SOFTWARE USER GUIDE

(Tutorials)

Computational Fluid x-Dynamics (CFxD)

(Version 2.03.4, March 17, 2026)

2026

Contents

	Nomenclature	2
	Getting started	4
I.	Basic tutorials	5
	1. Flow in the pipe (incompressible internal flow)	5
	2. Flow around a car (incompressible external flow)	23
	3. Flow in a duct with heated rods (flow with heat transfer)	31
	4. Buoyant flow in a closed box (natural convection)	38
	5. Free jet in open air (low Mach compressible flow)	43
	References	49

Nomenclature

Symbol	Dimension	Definition
C_i , c_i		model coefficients
c_p	$J/(kg\times K)$	specific heat capacity by constant pressure
g_i	m/s^2	gravity acceleration
Kn		Knudsen number
k	m^2/s^2	turbulent kinetic energy
l	m	characteristic length scale
M		Mach number
m	kg	mass
Nu		Nusselt number
P_k		turbulent production
Pr		Prandtl number
Pr_t		Prandtl turbulent number
p	Pa	static pressure
R	$J/(kg\times K)$	gas constant
Re		Reynolds number
Sc		Schmidt number
T	K	temperature
t	s	time
u, u_i	m/s	velocity vector/components
$\overline{u_i'u_j'}$	m^2/s^2	Reynolds stress tensor (components)
$\overline{u_j'\varphi'}$		scalar flux (components)
V	m^3	volume
x_i	m	cartesian coordinates

Greek letters

Symbol	Dimension	Definition
$\alpha_i, \beta_i, \delta_i, \gamma_i$		model coefficients
$\dot{\gamma}$		strain rate

Γ_φ		Molecular diffusion coefficient of a general scalar quantity, φ
δ_{ij}		Cartesian components of unit tensor (Kronecker delta)
ε	m^2/s^2	dissipation rate of turbulent kinetic energy
λ	$J/(s \times K \times m)$	thermal conductivity
φ		general scalar quantity
κ		Karman constant
μ, μ_t	$kg/(m \times s)$	dynamic molecular/turbulent viscosity
ν, ν_t	m^2/s	kinematic molecular/ turbulent viscosity
ρ	kg/m^3	density
σ	m^2	surface
σ_φ		turbulent Schmidt (or Prandtl) number for variable
τ	s	turbulent time scale
τ_{ij}		Reynolds stress tensor (components)
ω		specific dissipation rate

Abbreviations

Abbreviation	Definition
CAD	Computer-aided design
CDS	Central Difference Scheme
CFD	Computational Fluid Dynamics
CV	Control Volume
DNS	Direct Numerical Simulation
EVM	Eddy-Viscosity Model
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier-Stokes
RSM	Reynolds Stress Model
SIMPLE	Semi Implicit Method for Pressure Linked Equations
UDS	Upwind Difference Scheme
URANS	Unsteady RANS

Getting started

This document is an addition to Software User Guide (*Theory and general overview*). While the main guide explains the underlying physics, numerical methods, and software features, this document focuses on **practical, step-by-step tutorials** designed to help new users gain experience with CFxD.

The tutorials cover the complete workflow of a CFD simulation:

- Creating or importing geometry
- Generating a mesh
- Configuring physical models and boundary conditions
- Running the numerical solver
- Visualizing and interpreting results.

Each tutorial is organized to gradually introduce new concepts and tools in CFxD, allowing users to build confidence and familiarity with the software through hands-on examples.

These exercises are intended for:

- Students learning CFD fundamentals
- Engineers evaluating CFxD for practical use.

Throughout the tutorials, screenshots, tips, and explanations are provided to ensure that each step is easy to follow, even for users with limited or no CFD experience.

Chapter I. Basic tutorials

This chapter includes basic cases with steady incompressible flows with/without heat transfer.

1. Incompressible flow in a pipe

The configuration represents a pipe of 0.6 m diameter (D) with a 90° bend and with upstream and downstream straight sections of length 5D and 10D. The inlet velocity is 2 m/s and the fluid is air. We can verify that flow is turbulent and incompressible by calculating Re and M.

$$Re = \frac{D_0 \cdot u_0}{\nu} = \frac{0.6 \cdot 2}{1.46 \cdot 10^{-5}} \approx 82000.$$

The Reynolds number is approximately 82000, which is greater than 2300 (flow is turbulent in straight pipes if Re>2300 [1]), so flow is turbulent.

$$M = \frac{u}{a} = \frac{u}{\sqrt{kRT}} = \frac{2}{\sqrt{1.4 \cdot 287 \cdot 288}} = 0.006.$$

Mach number is much smaller than 0.3 [1], so compressibility effects can be neglected.

Start CFxD from Windows Start menu or from Desktop icon. The program will be started with the **Geometry** Tab open. The default project main file projectN.cfxd (N- number) and required folders will be generated automatically.

Geometry

There are the different ways to create a geometry. Here, the central curve, orthogonal circle surface, pull and combine operation will be used. Create the points to generate the center curve shown in Table 1.

Table 1: Point coordinates to create the center line.

Point	Coordinates	Comments
Pnt1	(0.0, 0.0, 0.0)	Start point of upstream pipe
Pnt2	(3.0, 0.0, 0.0)	End point of upstream pipe (Start point of arc)
Pnt3	(3.0, 0.6, 0.0)	Center of arc
Pnt4	(3.6, 0.6, 0.0)	Start point downstream pipe (End point of arc)
Pnt5	(3.6, 6.6, 0.0)	End point of downstream pipe

Point creation is shown in Figure 1. In the **Geometry creation** panel, select **Create point** (1). Enter the point's coordinates in **Coordinates input** panel (2). The empty input will be automatically recognized as 0.0. Generate a point with **Create point** button (3). Any created geometry will appear in **Geometrical entities** tree (4) and will be displayed in the **Geometry view** window (5). Helpful hints are shown in the **status bar** at the bottom (6).

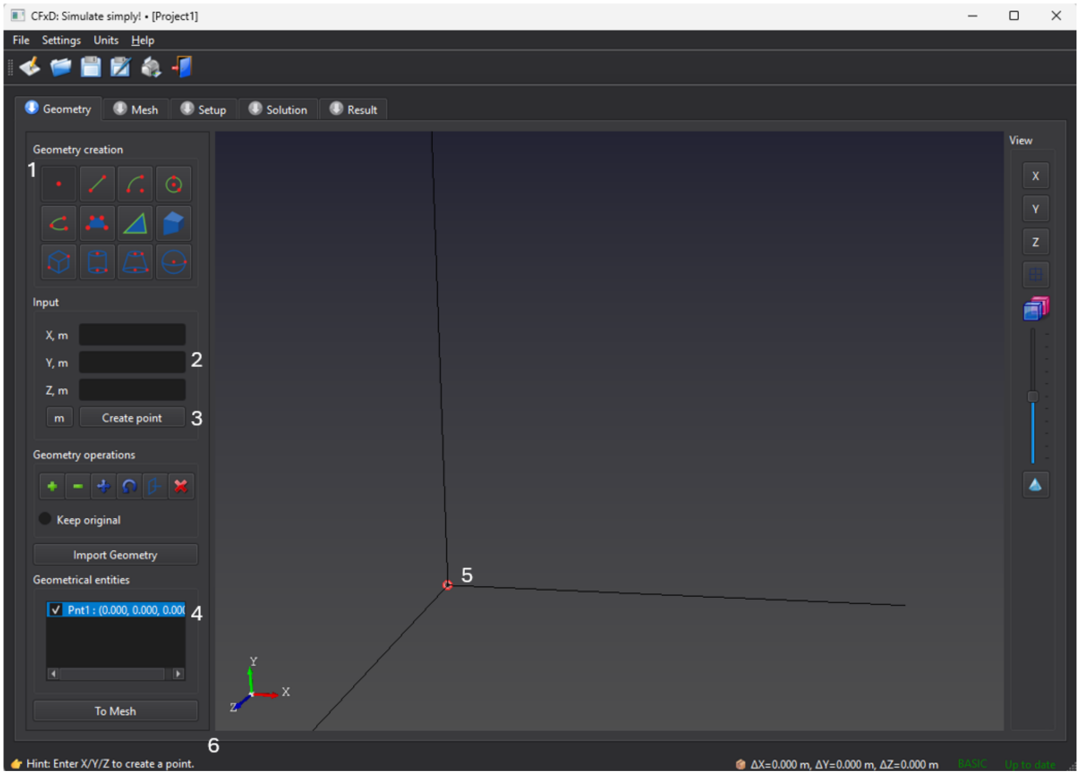

Figure 1. Point creation.

All five points are shown in Figure 2.

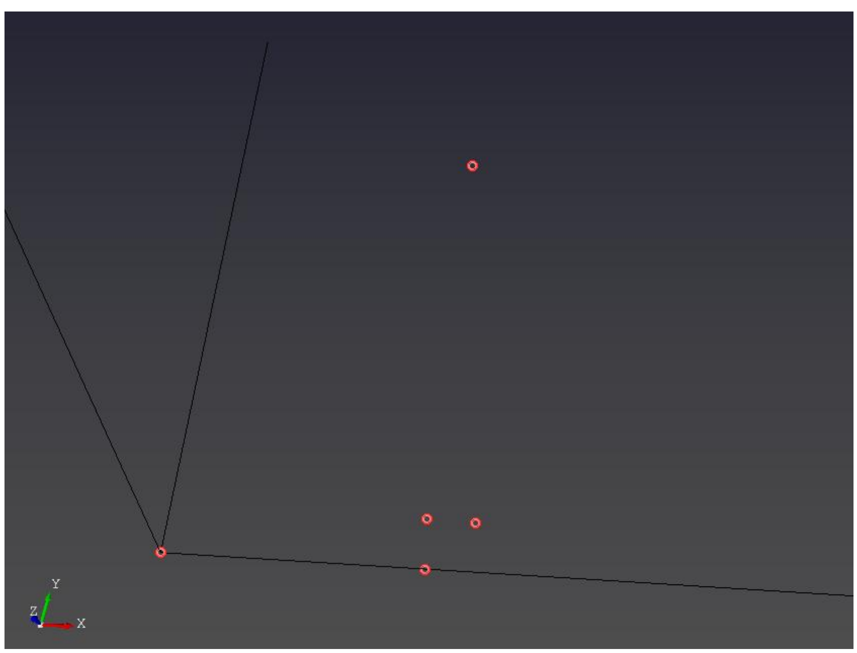

Figure 2. Five created points.

The next step is to create lines by selecting two points and arc by selecting three points with left mouse button (LMB). For an arc, the selection order is center point → start arc point → end arc point.
The hint on the bottom of screen can be used for guidance. The created two lines and arc are shown in Figure 3. Any selection can be discarded with LMB on empty screen.

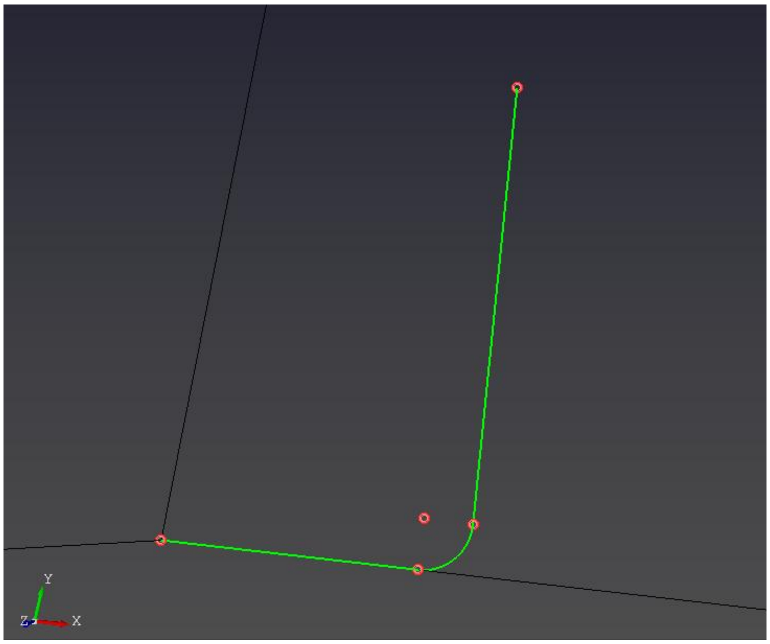

Figure 3. Created lines and arc.

Create two additional points (Table 2) to define a circle lying in the YZ plane. The points can be generated without switching back to the **Create point** tool. As the center point the Pnt1 will be used.

Table 2: Points coordinate for the circle.

Point	Coordinates	Comments
Pnt6	(0.0, 0.3, 0.0)	First point on circle
Pnt7	(0.0, 0.0, 0.3)	Second point on circle

The new points are shown in Figure 4.

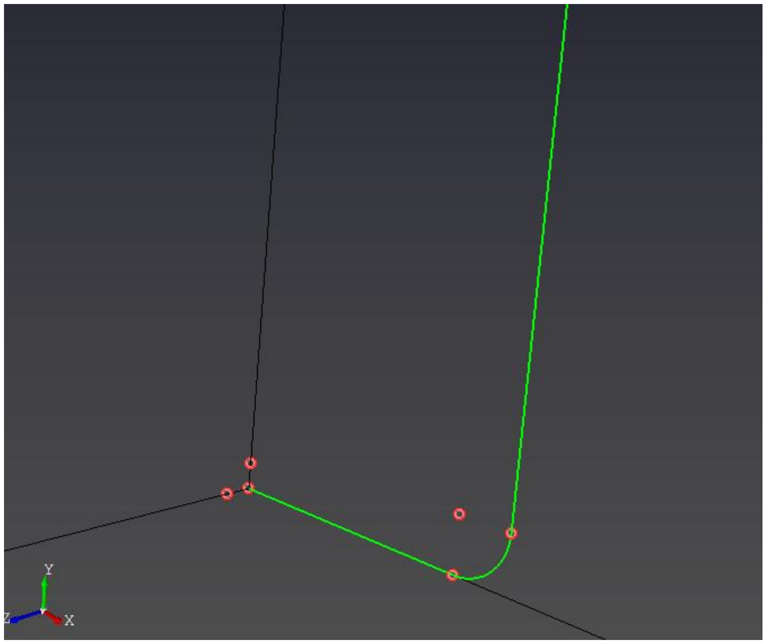

Figure 4. Points for circle creation.

Click on **Create circle** tool  and select three points in the following order: center of circle → first point on circle → second point on circle. The created circle is shown in Figure 5.

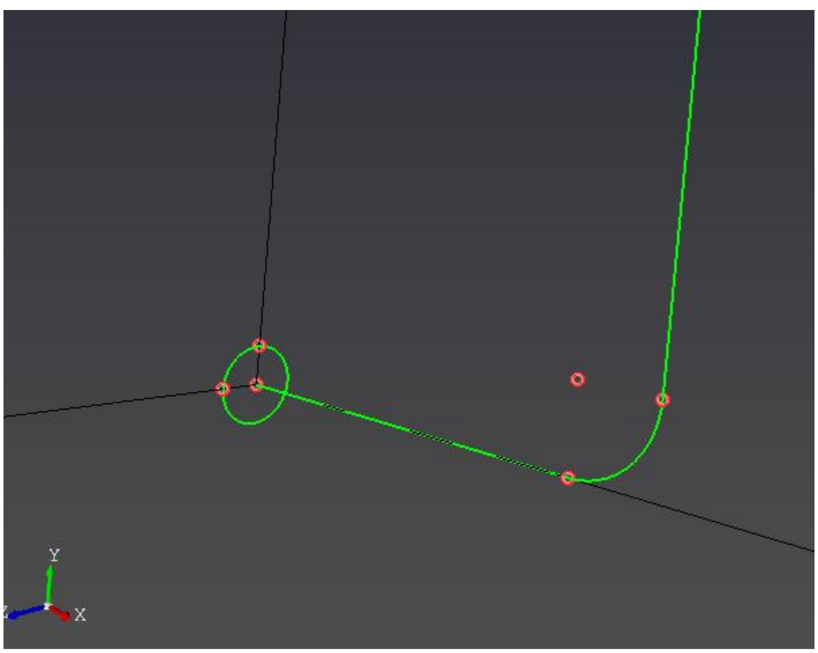

Figure 5. Circle creation.

Click the **Create surface** tool and select the circle. Since the circle forms a closed loop, a surface will be created (Figure 6).

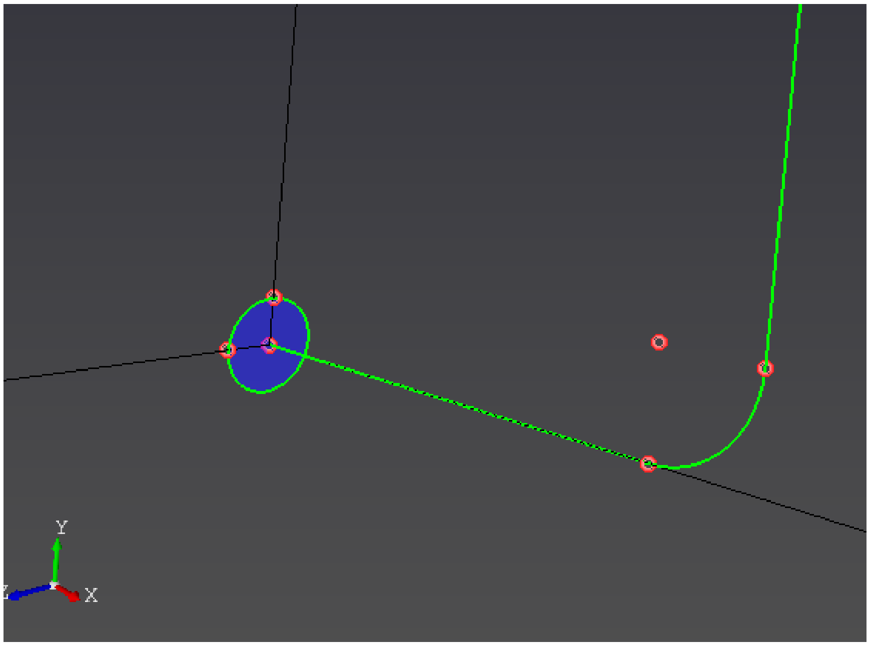

Figure 6. Circular surface generation.

The current geometry can be saved with **Save** icon in the toolbar, from the menu or with Ctrl+S keys.

Activate a **Pull** tool in the **Geometry operations** panel. Select the circular surface and the line attached to it. A body will be created automatically (Figure 7).

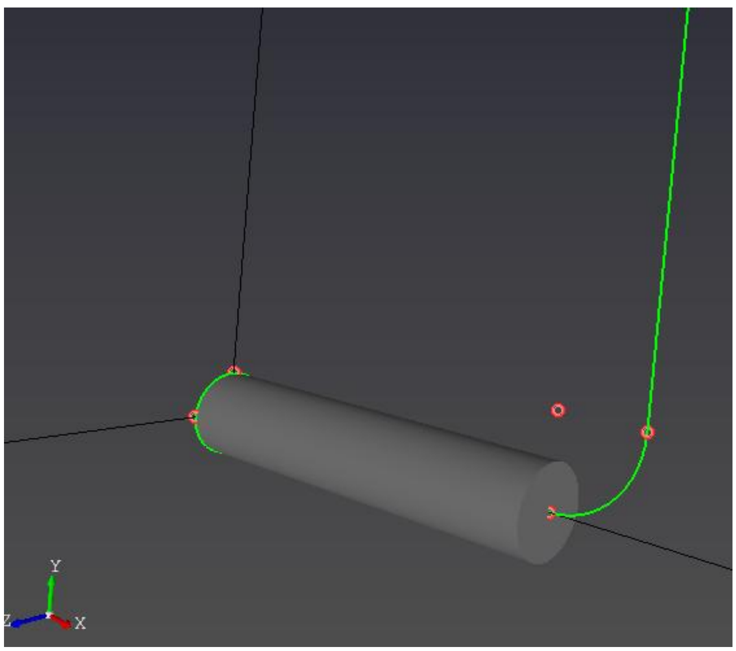

Figure 7. Cylinder creation using Pull operation.

Repeat the same procedure for bend section and the downstream pipe (Figure 8).

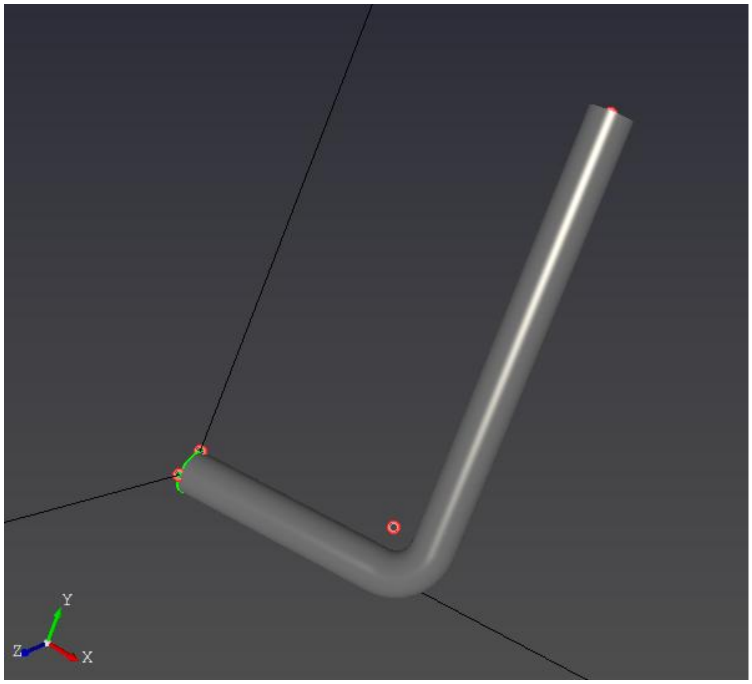

Figure 8. Upstream, bend and downstream pipes.

At this stage, there are three separate bodies. Click on Boolean operation: unite , keep **Keep original** unchecked, and select the upstream and bend bodies. A new united body will be created, and the originals bodies will be removed. Repeat the same operation with newly united body and the downstream pipe (Figure 9).

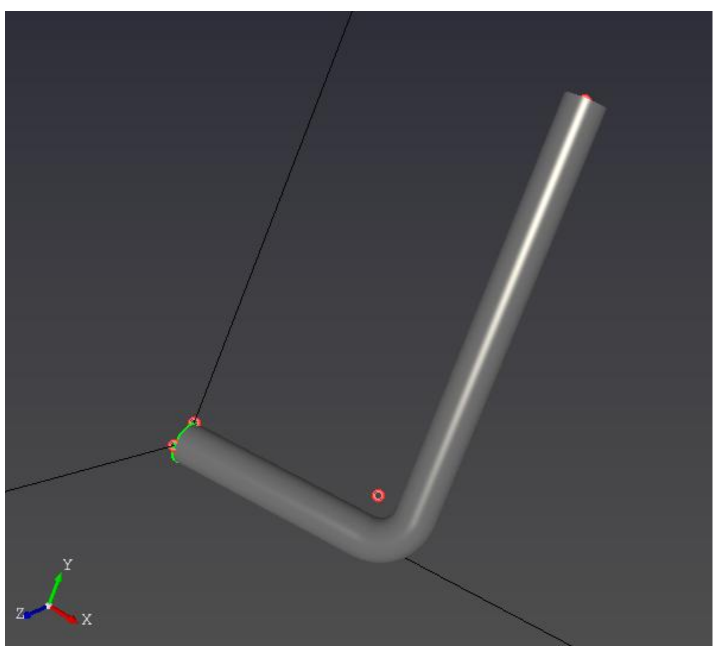

Figure 9. Final pipe geometry.

Enable **Face naming** mode by right-clicking RMB in the **Geometry view** window. The mouse cursor will change to a crosshair. Right-click the face at the inlet side of the pipe and name it **inlet** in the dialog (Figure 10). The assigned names will appear in the **Geometrical entities** tree.

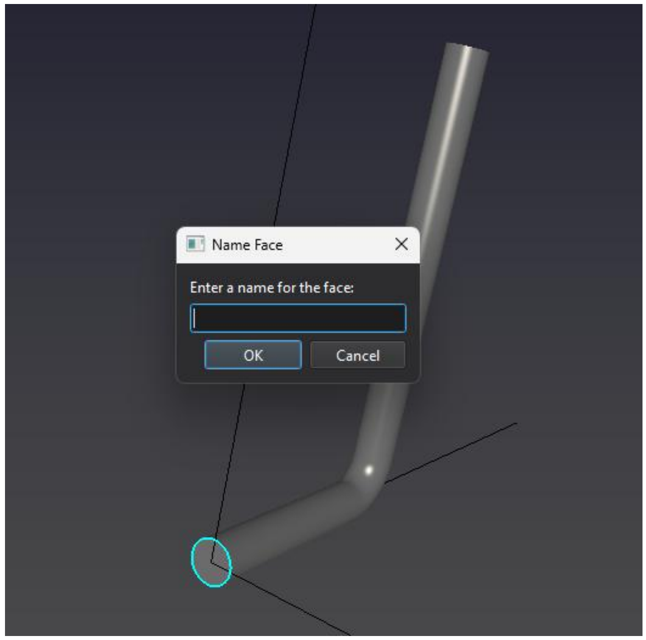

Figure 10. Face naming.

Similarly, name the exit face **outlet**. All remaining faces will be automatically assigned as **wall**. Click To Mesh to proceed to the next stage. The program will save all required files and the current state.

Mesh

The **Mesh** generation tab is shown in Figure 11.

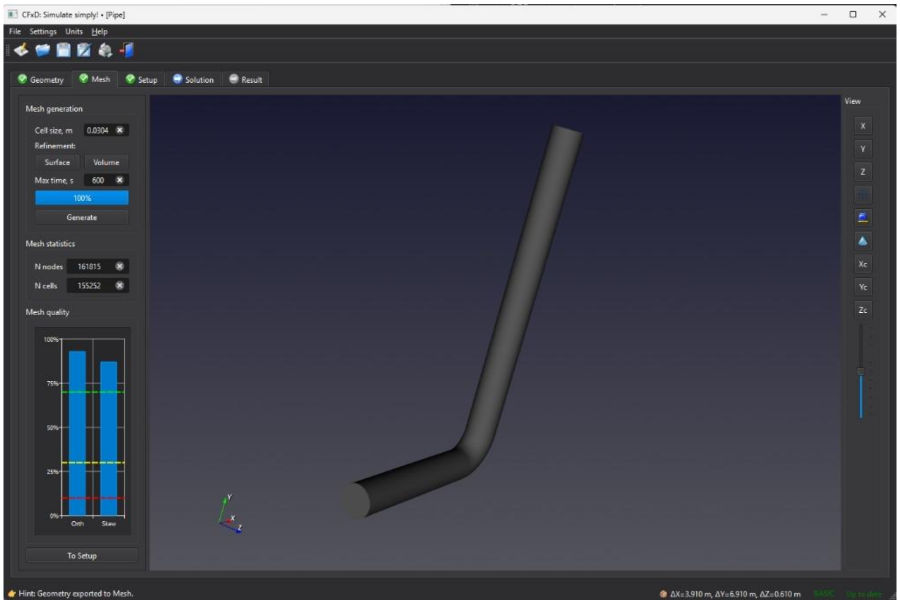

Figure 11. Mesh generation tab.

Keep the default pre-calculated **Cell size** (0.0304) and click on **Surface** or use RMB in **Mesh Viewer** to call Surface refinement dialog. Select **wall**, keep the default **Local cell** and enable **Layers** to generate prism (boundary-layer) layers near the walls. Use **Preview** to see the selected geometry and cell size (Figure 12). After it **Add** wall. Finally, **Apply and Close** dialog.

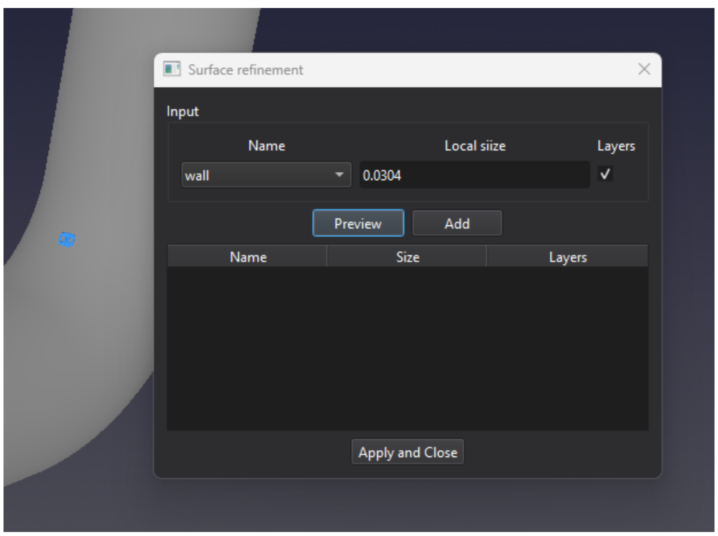

Figure 12. Set layers for wall.

Keep the default **Maximum time** 600 s. In this case the mesh generation process will stop automatically when the time limit is reached. Finally, click **Generate**. The generated mesh is shown in Figure 13.

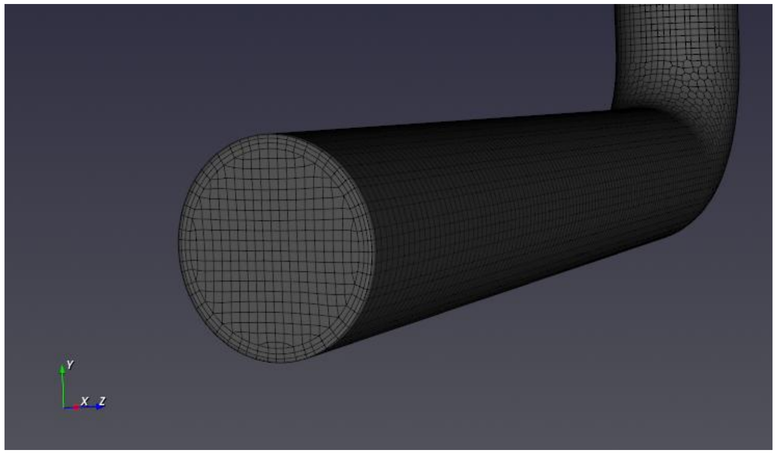

Figure 13. Generated mesh.

The number of generated nodes and cells can be checked in **Mesh statistics** panel. Mesh quality is shown as histograms of orthogonality (**Orth**) and skewness (**Skew**), along with quality thresholds (Figure 14).

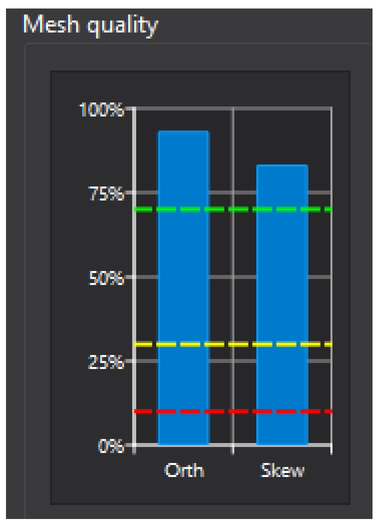

Figure 14. Mesh quality graph.

To inspect the mesh inside pipe, use **Section cut** buttons Xc, Yc, Zc on the right side of the window (Figure 15). The cut position can be adjusted using the slider.

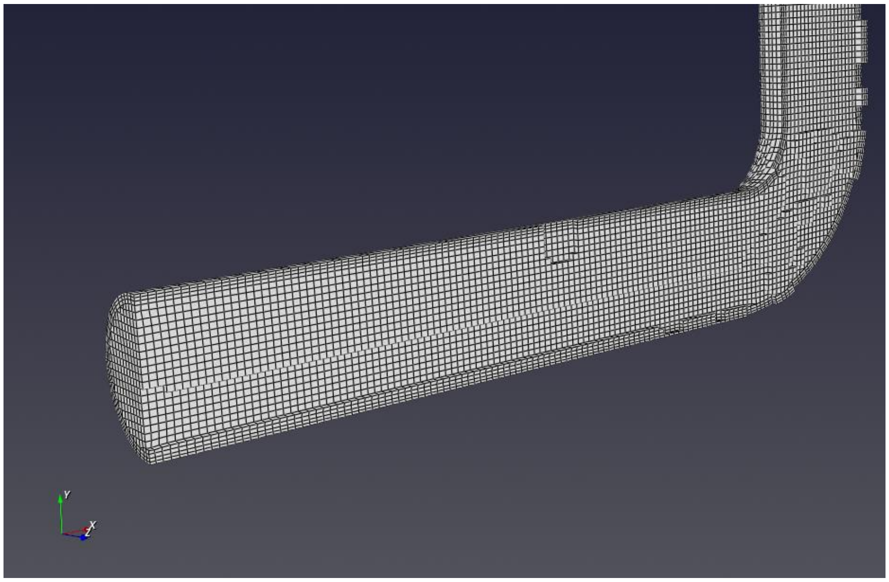

Figure 15. Mesh cut in Zc-direction.

Small visual artefacts may appear on the cut plane due to imperfect alignment between the cut and the mesh cells.

Click **To Setup** to proceed to the next stage.

Setup

The **Setup** tab is shown in Figure 16.

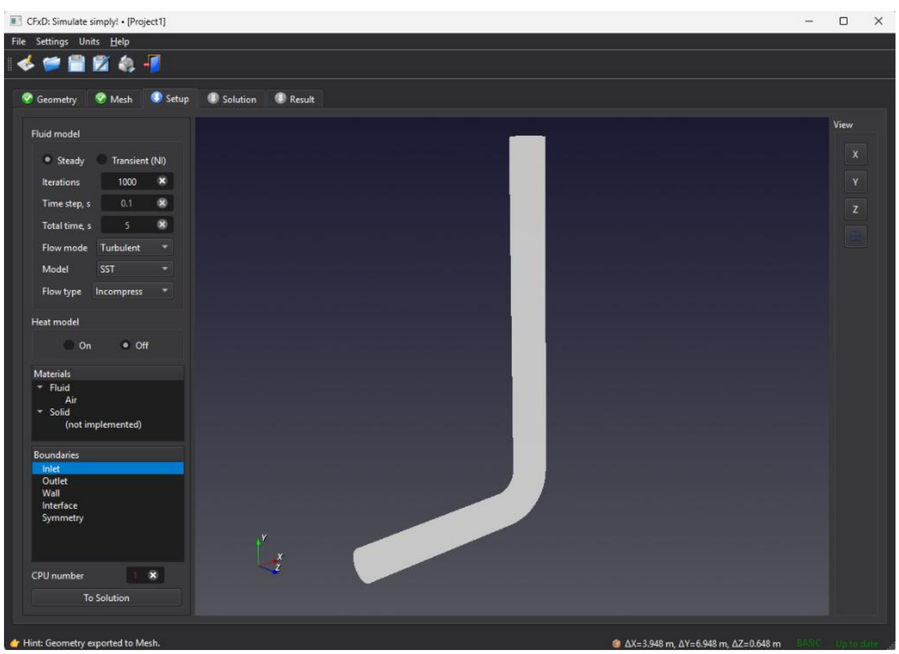

Figure 16. Setup Tab.

Keep the default settings in the **Fluid model** panel:

- **Steady** flow
- **1000** iterations
- **Turbulent** flow mode
- **SST** turbulent model
- Incompressible (**Incompress**) flow.

There is no heat transfer in this case so set **Heat** to **Off**. Keep the default **Fluid** in the **Material** tree: **Air**. In the **Boundaries** tree, click on **Inlet**. In the **Inlet** dialog, set:

- **x-Velocity**: 2 m/s
- **Length scale**: 0.6 m (pipe diameter)
- all other inputs are default.

Click **Apply** to save the changes (Figure 17).

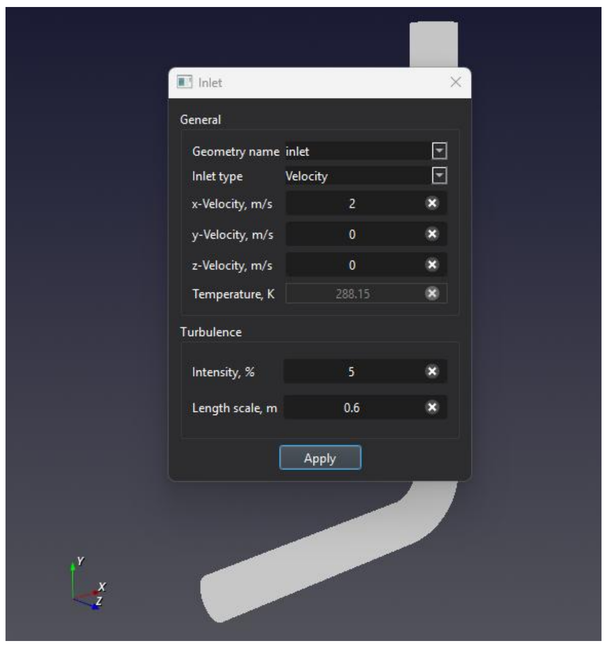

Figure 17. Setup of inlet boundary conditions.

Similarly, open the dialogs for **Outlet** and **Wall** and keep the default parameters. Because the faces were named **inlet** and **outlet** in the **Geometry** stage, the boundary-condition dialogs are automatically linked to the correct patches. After assigning the boundary conditions, the faces are color-coded in **Mesh view** window: **green** for **inlet**, **blue** for **outlet** and **dark gray** for **wall** (Figure 18).

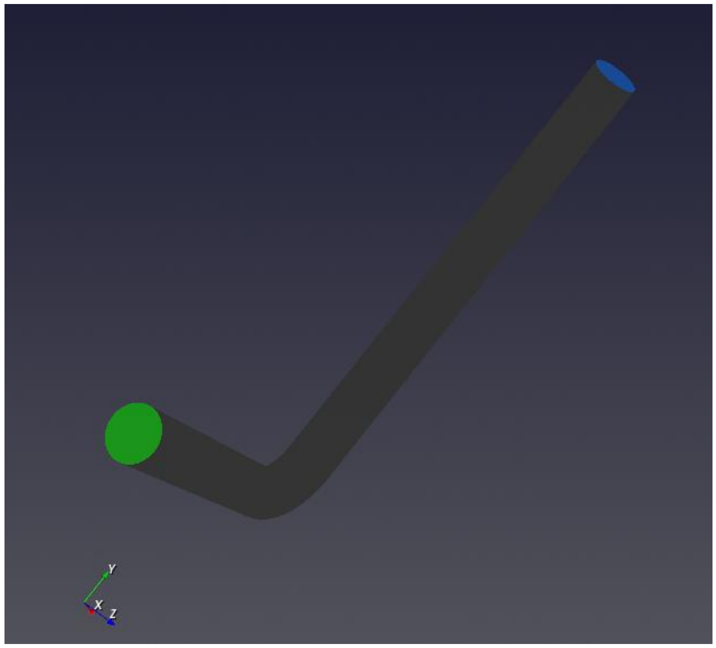

Figure 18. Final setup boundaries.

Keep the default number of CPU cores (1). Click **To Solution** to proceed.

Solution

The **Solution** tab is shown in Figure 19.

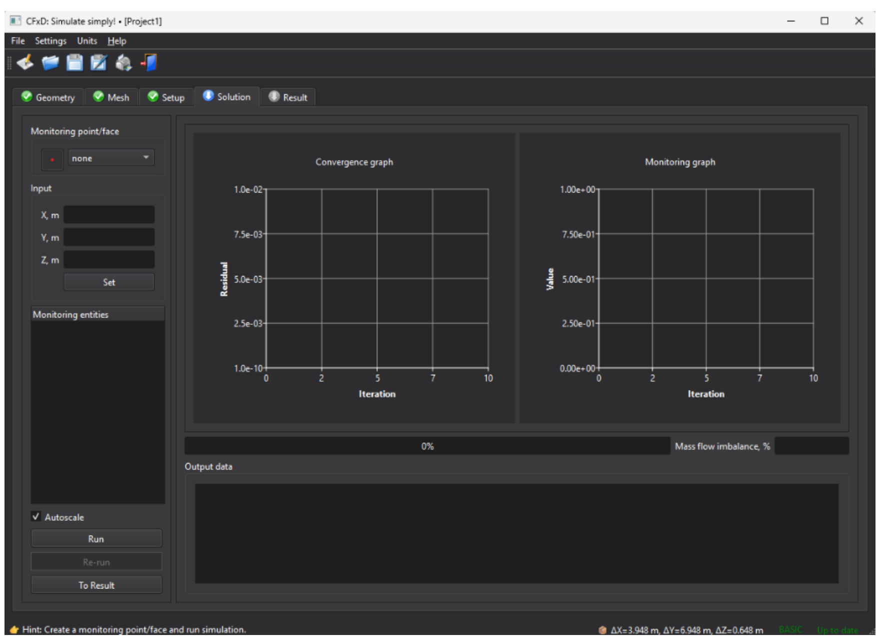

Figure 19. Solution tab.

In **Monitoring point/face** panel, select **Outlet** (instead of **none)**. The outlet entity will appear in the **Monitoring entities** tree. Click **Outlet** to open the variable selection dialog. Select **Umag** and click **Apply** (Figure 20).

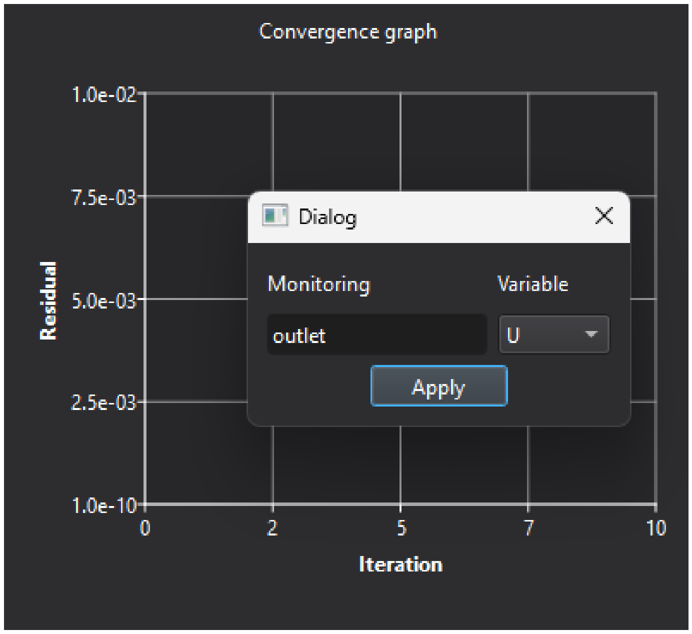

Figure 20. Setup of monitoring face at outlet.

In **Input** panel, set X = 3.6, Y = 6.0, Z = 0.0 and click Set to create a monitoring point. Click the new point in the **Monitoring entities** panel, assign the monitoring variable **Umag**, and confirm with **Apply**. Start the simulation by clicking **Run**. A converged solution is shown in Figure 21.

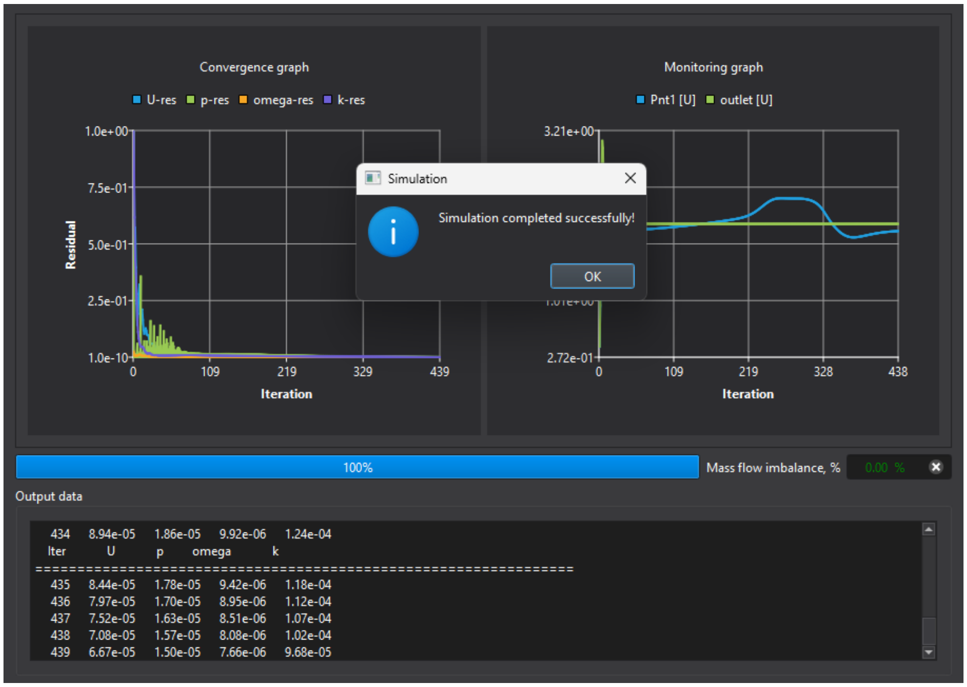

Figure 21. The completed simulation.

Residual values can be inspected in the **Output data** panel and on the convergence graph. Click **To Result** to proceed to the Result tab.

Result

The **Result** tab is shown in Figure 22.

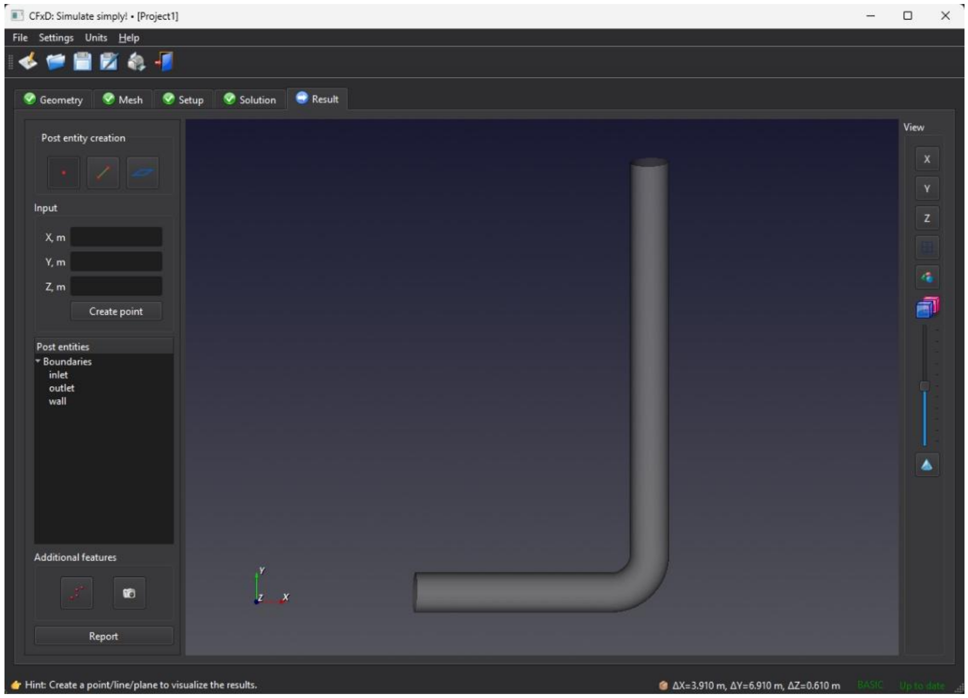

Figure 22. Result tab.

In the **Post entity creation** panel, click on **Create point** button (active by default). Create a point at X = 3.6, Y = 6.0, Z = 0.0 using the **Input** panel. The point will appear in the **Post entities** tree and **Result view** window. Click on point **Post entities** tree to open the dialog and view values at that location. The three-velocity components at the selected point are shown in Figure 23.

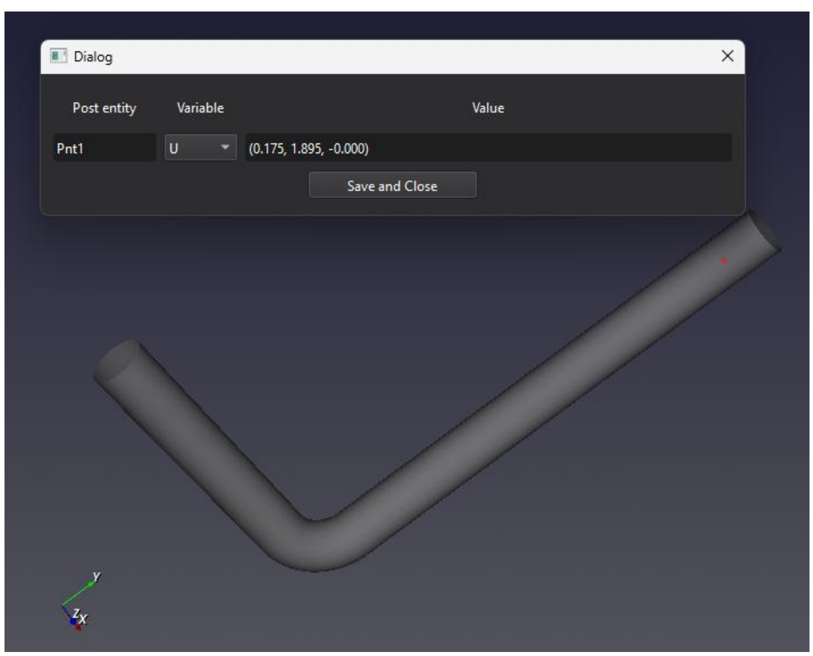

Figure 23. Velocity components value at selected point.

Click **Create line** button in the **Post entity creation** panel to generate a line plot. Define the line using two points:

- Point 1: X = 2.9, Y = 6.0, Z = 0.0
- Point 2: X = 4.3, Y = 6.0, Z = 0.0.

The line will appear in the **Result view** window and in the **Post entities** tree. Click the line in the tree to open its dialog. Select velocity **Umag** and click **Show chart** (Figure 24).

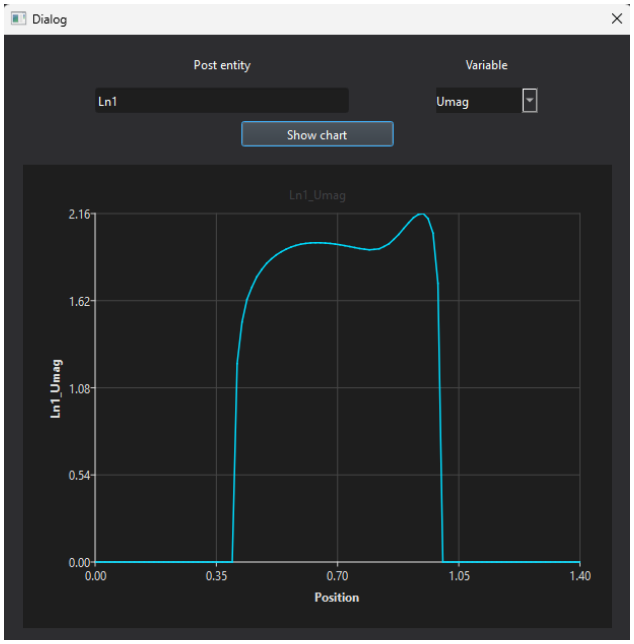

Figure 24. Velocity profile at selected line.

Create a plane using the **Plane** button. Keep the default coordinates to create XY plane at Z = 0. In the plane dialog, select velocity **Umag** and click on **Close and show** to display the contour plot in the **Result view** window (Figure 25).

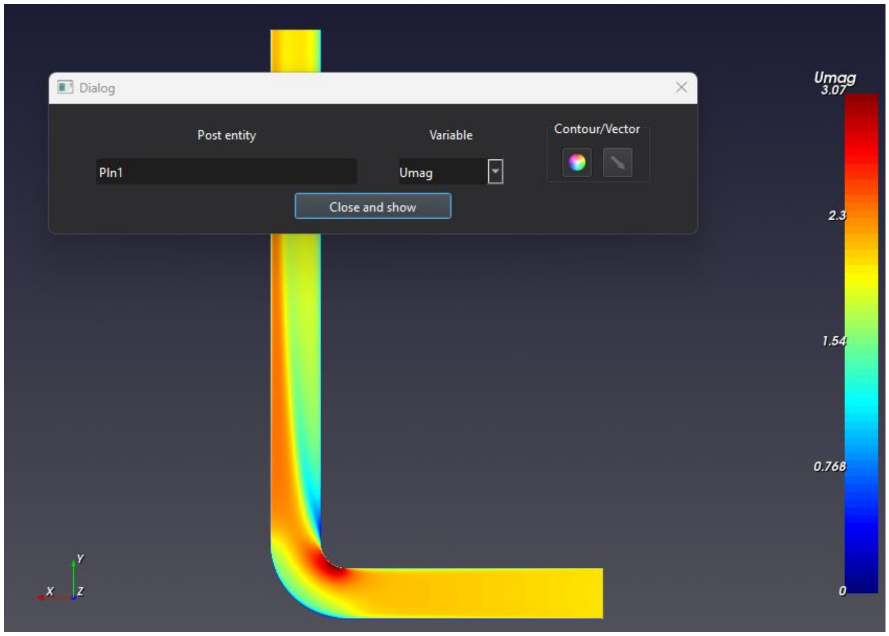

Figure 25. Velocity contour on the mid-plane (XY, Z=0).

Boundary patches are added automatically to the **Post entities** tree. Click **wall**, then in the dialog select **pAbs (absolute pressure)**. The pressure distribution on the wall is shown in Figure 26.

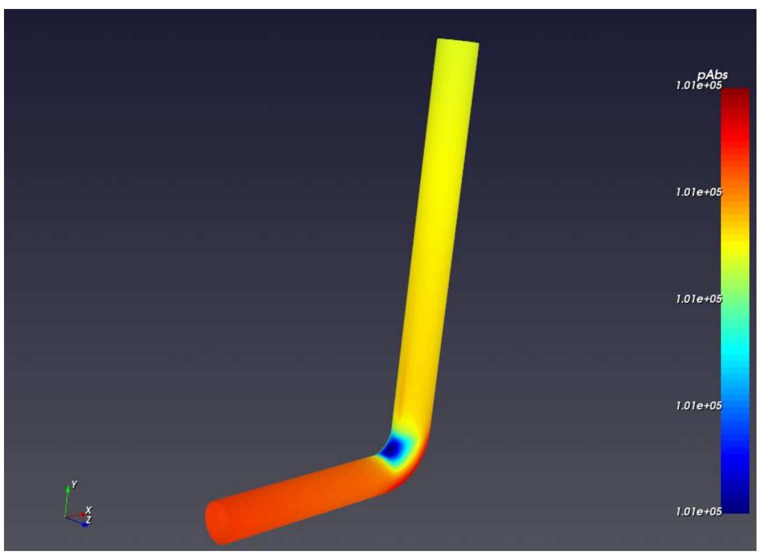

Figure 26. Absolute pressure on the wall.

Finally, click **Report**. A PDF report will be generated automatically (Figure 27) and saved in the **Result** folder. The project will be saved as well.

CFxD Simulation Report

Project: Project1

Path: C:\CFxD\Projects\Project1

Date: 2026-01-30 15:52:50

Geometry

Statistics

- **Solids:** 1
- **Faces:** 6
- **Edges:** 11
- **Vertices:** 18

Mesh

Statistics

- **Cells:** 102338
- **Nodes:** 107704

Quality

- **Orthogonality (≥ threshold):** 93 %
- **Skewness (≤ threshold):** 83 %

Figure 27. Generated report.

2. Flow around car

This tutorial uses a prepared simplified car model in **STEP** format. For external aerodynamics, the car must be placed inside a rectangular **computational domain**. The car speed is **200 km/h** (≈ **55 m/s**). Start CFxD from Windows menu or desktop icon.

Geometry

Import the **car** CAD model (STEP format). In CFxD, click **Import Geometry** (Figure 28), locate **car.stp** and open it.

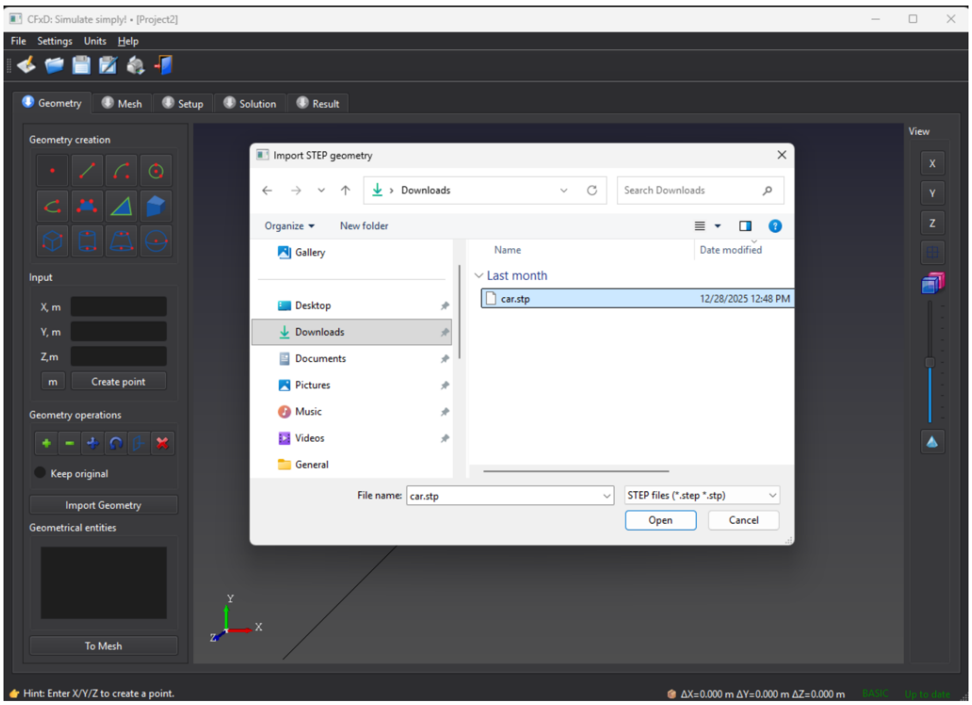

Figure 28. Importing geometry.

To model flow around a car, create a box enclosure. Use the **Box** tool from **Geometry creation** panel. Create two corner points (Table 3). These points define opposite corners of the enclosure (Figure 29).

Table 3: Points required to generate a domain.

Point	Coordinates	Comments
Pnt1	(9.25, 5.0, 0.0)	First corner point of the box
Pnt2	(-6.25, 0.0, 5.0)	Second corner point of the box

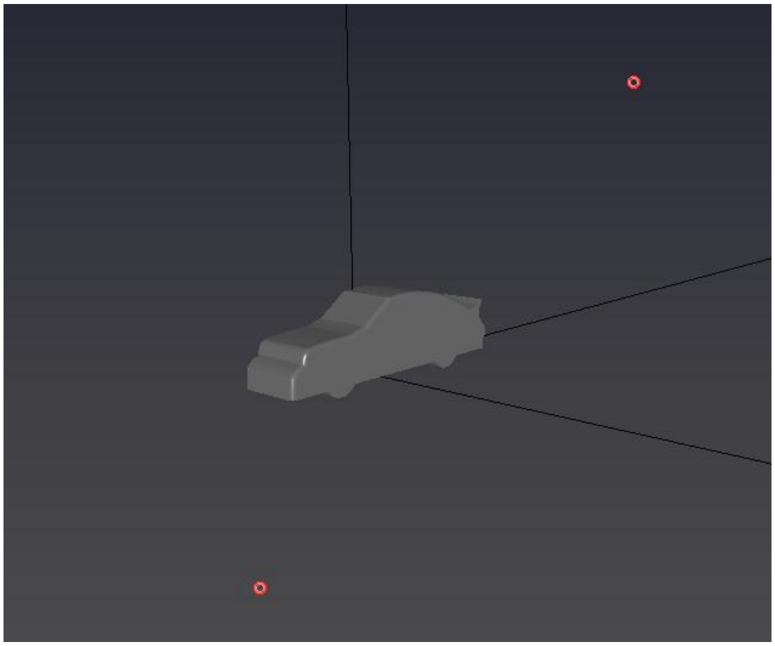

Figure 29. Corner points of domain.

Click **Box** in the **Geometry creation** panel and select the two points (Figure 30).

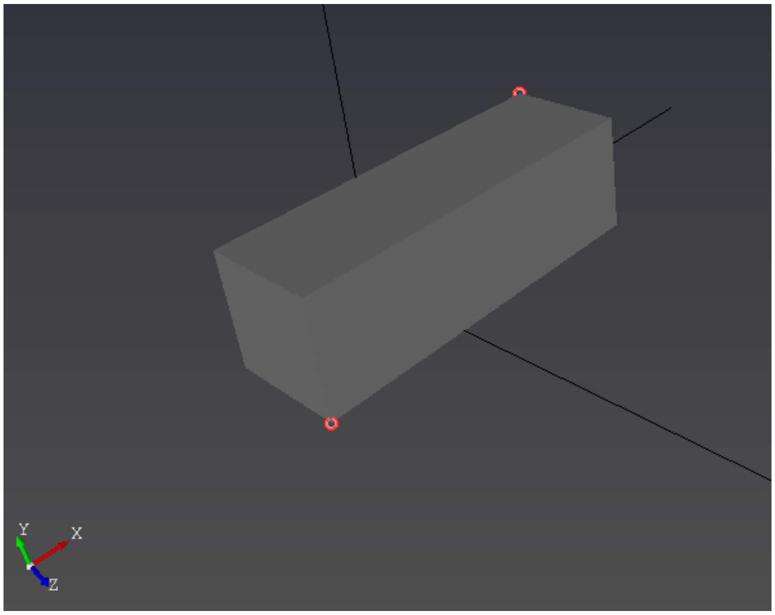

Figure 30. Created domain.

Next, subtract the car body from the enclosure so that only the **fluid domain** remains. In the **Geometry Operations** panel, click **Subtract**. Select the **enclosure** first. Then hide the enclosure in the **Geometrical entities** tree by unchecking its visibility box. With the enclosure hidden, select the **car body** to subtract it from the enclosure. The resulting body is shown in Figure 31.

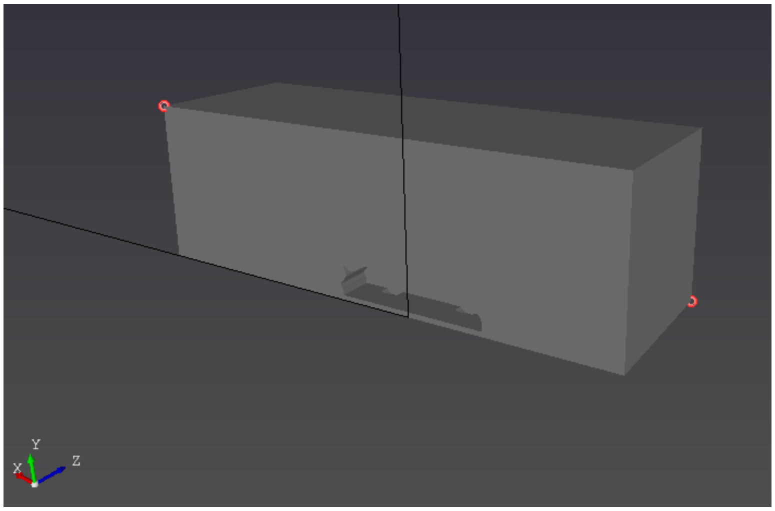

Figure 31. Final fluid domain.

Activate naming mode with **RMB** on an empty area in the graphical window. Name the **front face** of the domain **inlet** (Figure 32).

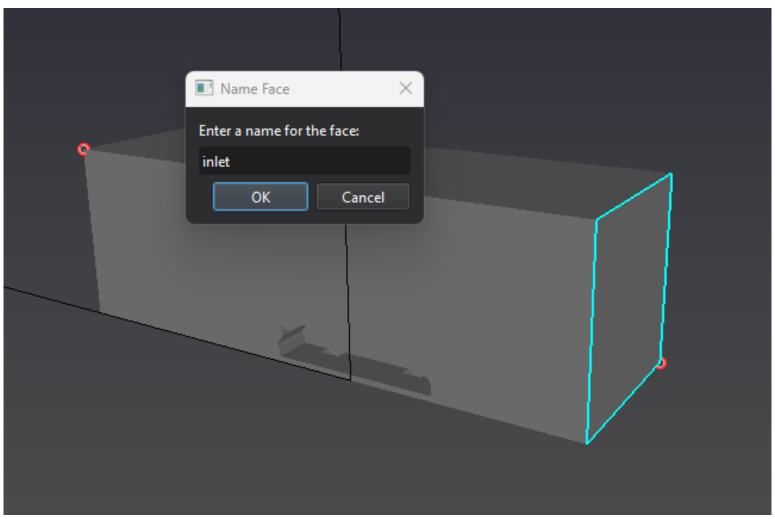

Figure 32. Naming the front face "inlet".

Name the **center symmetry face** as **symmetry-center**. Name the **top** and **side** faces as **symmetry**. To name multiple faces at once, hold **Ctrl** and select faces with **RMB**. You can still rotate the model using **LMB**. Releasing **Ctrl** opens the naming dialog (Figure 33). Two separately named symmetry faces are only required for post-processing.

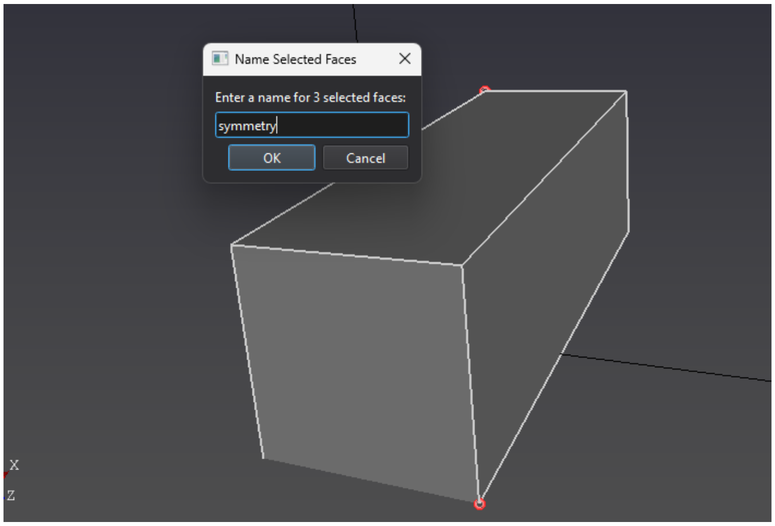

Figure 33. "Symmetry" name for the top and side faces.

Finally, name the rear face **outlet** (Figure 34).

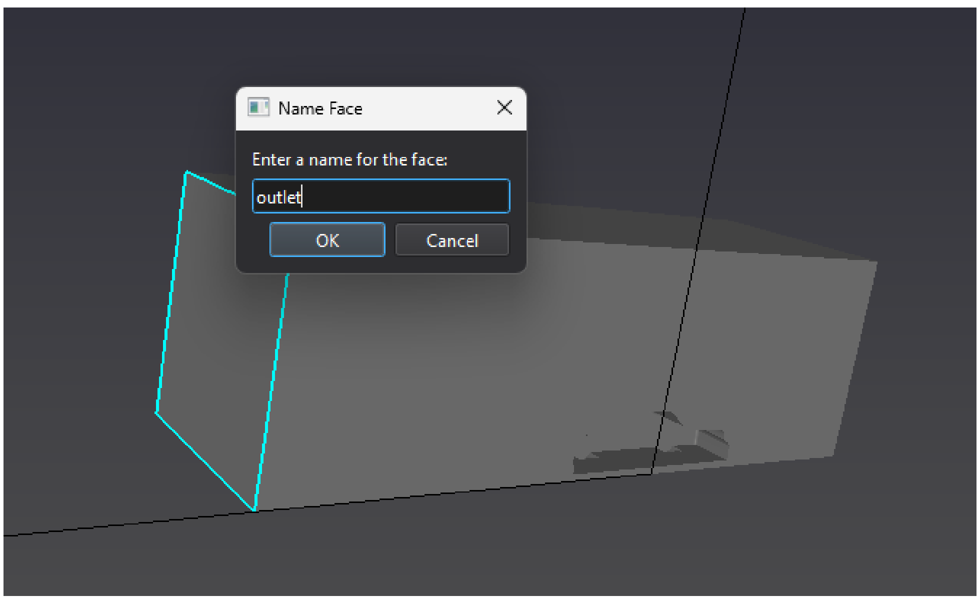

Figure 34. Naming the back face "outlet".

The remain face (ground) will be automatically named **wall**. Click **To Mesh** to proceed.

Mesh

Set **Refinement** level = 1. Keep the remaining default mesh settings (**Cell size**: 0.156, **Wall treatment**: no, **Max time**: 600) and click **Generate**. The generated mesh shown in (Figure 35).

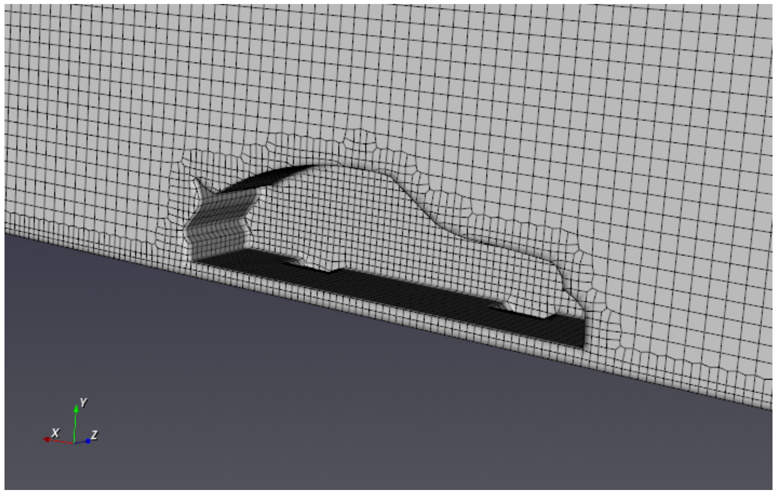

Figure 35. Generated mesh.

Click **To Setup** to continue.

Setup

Keep the default **Fluid model** settings. Ensure **Heat transfer** = Off and the material is **Air**. In the **Boundaries** tree click **inlet** and set **x-Velocity** = 55 m/s, and **Length scale** = 5 (Figure 36).

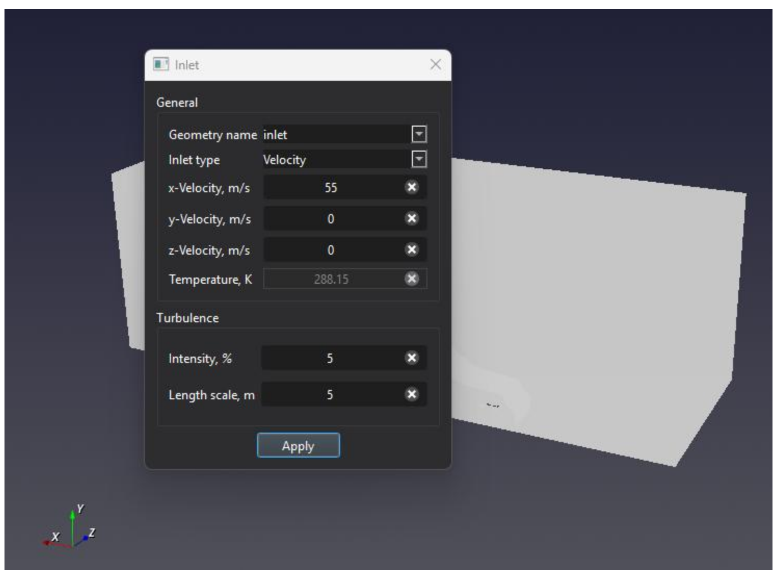

Figure 36. Inlet boundary conditions.

For **Outlet**, **Wall** and **Symmetry,** keep default parameters. Since two symmetry patches (**symmetry** and **symmetry-center**) were created, apply symmetry settings to both. All boundary conditions are shown in Figure 37.

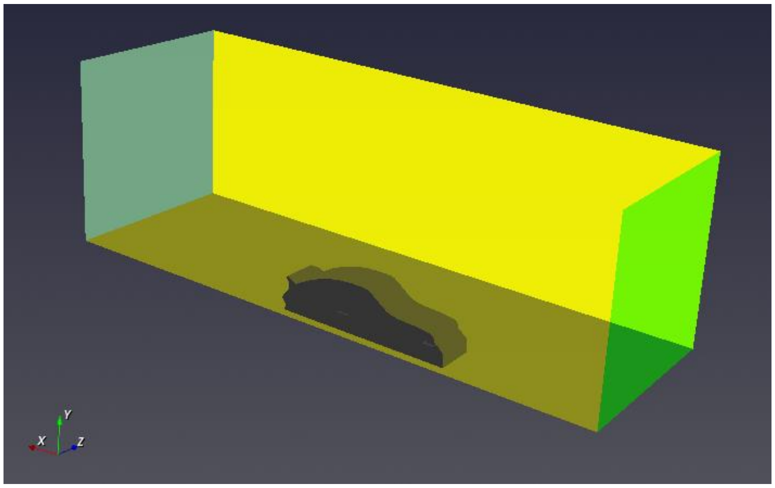

Figure 37. Boundary conditions.

Click **To Solution**.

Solution

Create a monitoring point using the coordinates in Table 4. Then click the point in the **Monitoring entities** tree and assign the variable **Umag** or **Ux**.

Table 4: A monitoring point.

Point	Coordinates	Comments
Pnt1	(3.0, 0.5, 0.0)	Monitor point behind the car

Click **Run** to start the simulation. When finished (Figure 38), click **To Result**.

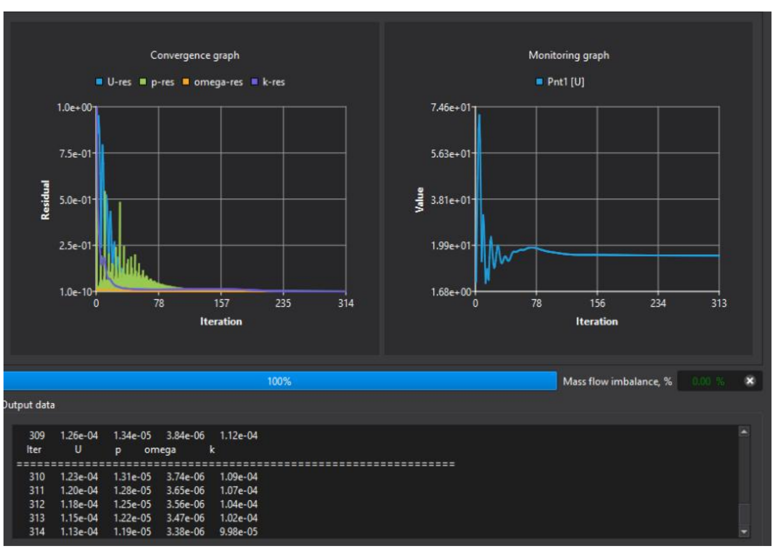

Figure 38. Finished simulation.

Result

Create a plane at y = 0.5 and display **Ux** on it. In the **Post entities** tree, expand the **Boundaries** category and click **symmetry-center**. In the dialog, select **Ux** (velocity x-component). Both contours will be displayed together (Figure 39).

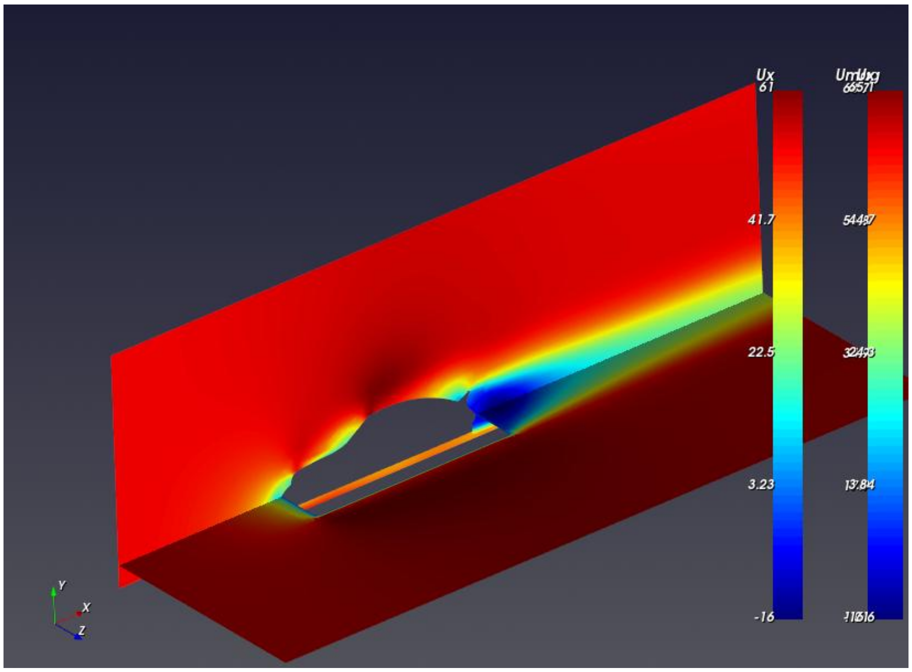

Figure 39. Two velocity contours.

Click **symmetry-center** again and select **U** in the dialog. It will create automatically a vector plot (Figure 40).

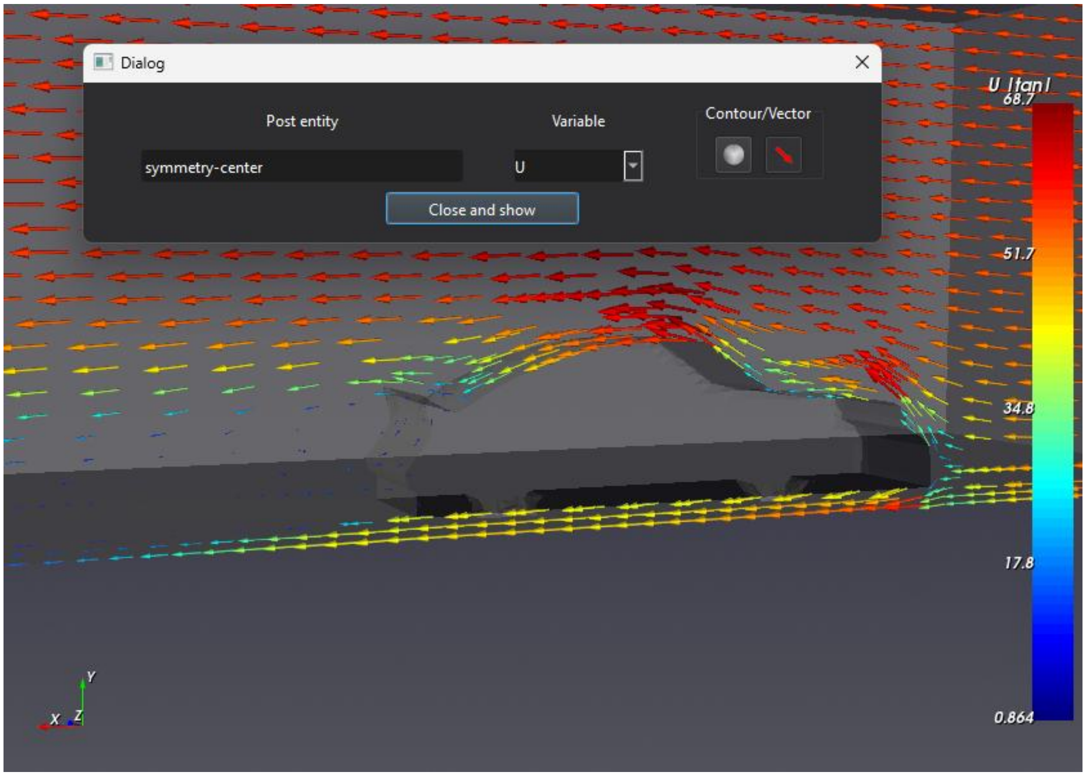

Figure 40. Vector plot on central symmetry plane.

Finally, click **Report** to generate the PDF report.

3. Flow in a duct with heated rods

This case models airflow through a rectangular duct with three heated rods. The duct dimensions are 100 mm (height) × 500 mm (width) × 1500 mm (length). The inlet velocity is 1 m/s and the air is fluid. Start CFxD from the Windows menu or the desktop icon.

Geometry

Create the points required to build the geometry. The points coordinates are listed in Table 5. Points Pnt 1, Pnt2 define the duct, and points Pnt3 - Pnt5 define one rod (cylinder). The units are in mm. In the **Input** panel, use **Unit** button to switch to mm before entering coordinates.

Table 5: Point coordinates (mm).

Point	Coordinates	Comments
Pnt1	(0, 0, 0)	First corner point of the box
Pnt2	(1500, 400, 100)	Second corner point of the box
Pnt3	(400, 100, 0)	Center point of rod circle
Pnt4	(425, 100, 0)	Radius point (25 mm)
Pnt5	(400, 100, 100)	Height point (100 mm)

The created points are shown in Figure 41.

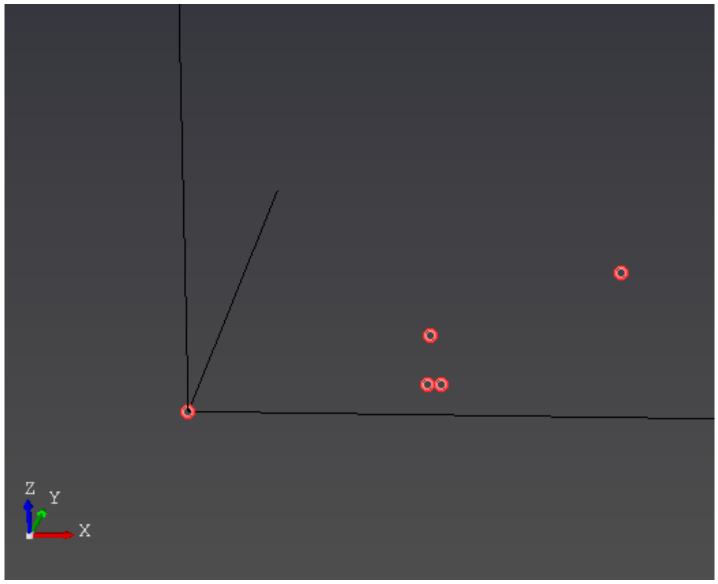

Figure 41. Created points.

Create a cylinder using Pnt3-Pnt5 (Figure 42).

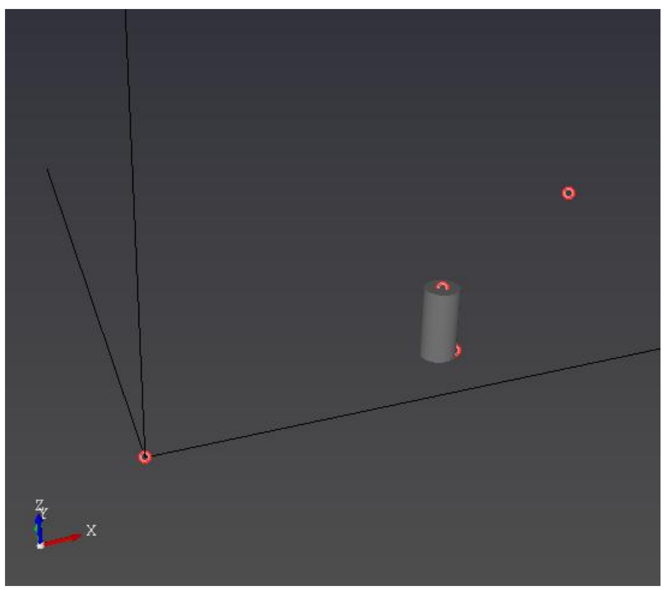

Figure 42. Created cylinder.

Select **Translate** operation, enable **Keep original**, and copy the cylinder to the new positions +100 mm and +200 mm in the Y direction (Figure 43).

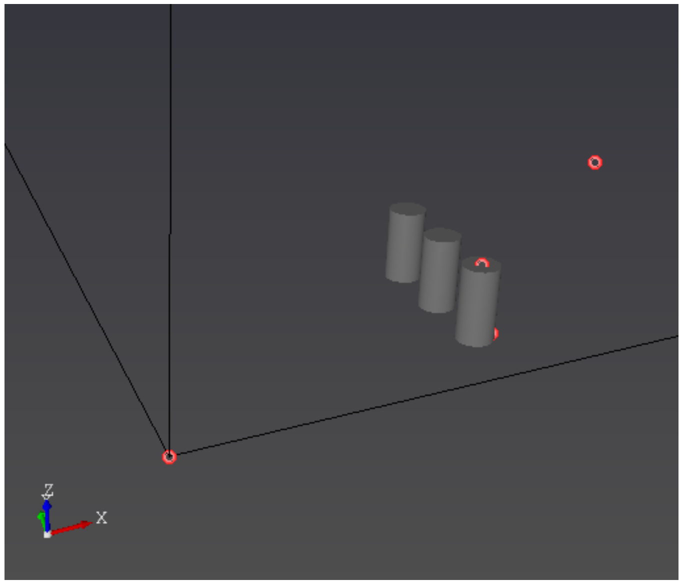

Figure 43. Cylinders created using translation.

Create a box (duct) using Pnt1 and Pnt2, then subtract the three cylinders from the box with unchecked **Keep original** (Figure 44).

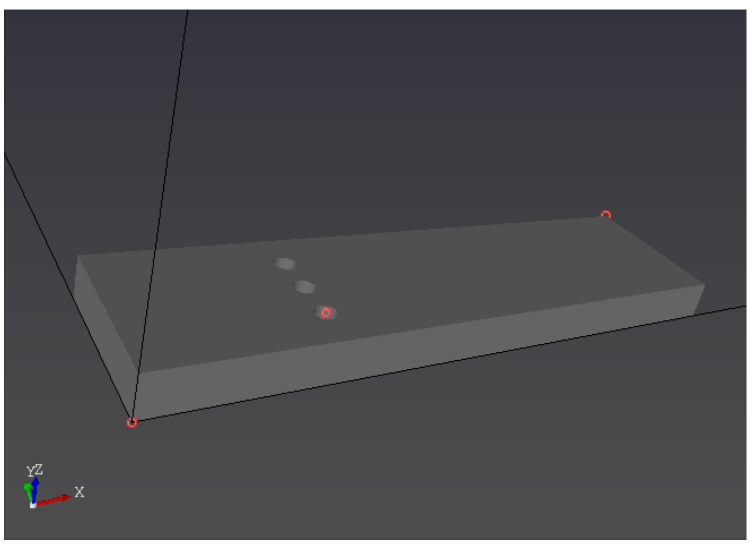

Figure 44. Final body.

Activate **Naming** mode with RMB. Name faces as follows:

- front face: **inlet**
- back face: **outlet**
- two sides: **wall**
- top and bottom: **symmetry**
- cylinders: **wall-hot**.

Use Ctrl+RMB for multi-selection. Releasing of Ctrl opens the naming dialog (Figure 45).

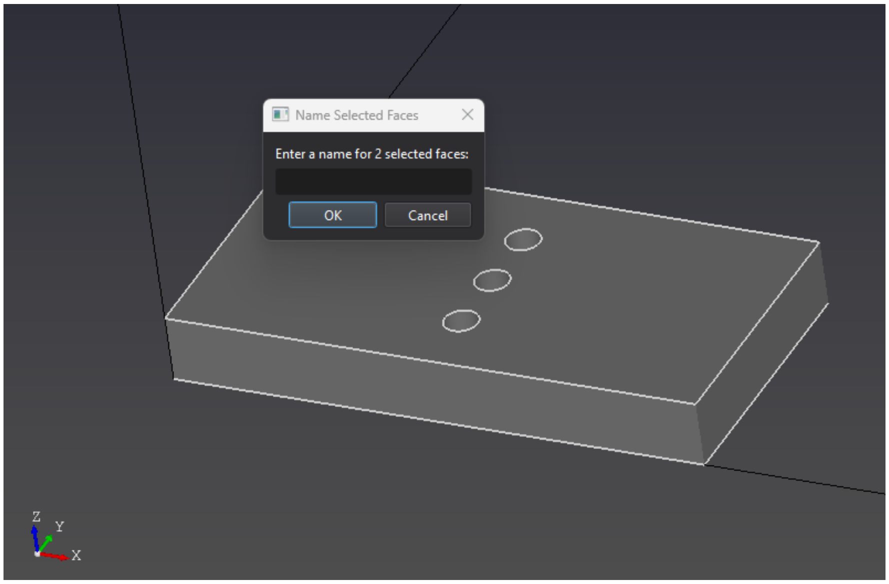

Figure 45. Naming "symmetry" for top and bottom faces.

Click **To mesh** step to proceed.

Mesh

Keep the default **Cell size**, set **Refinement** level = 1, and enable **Wall treatment**. Click **Generate**. Inspect the mesh with **Mesh cut**. An example cut in Y direction is shown in Figure 46. Also check the mesh quality plots before proceeding.

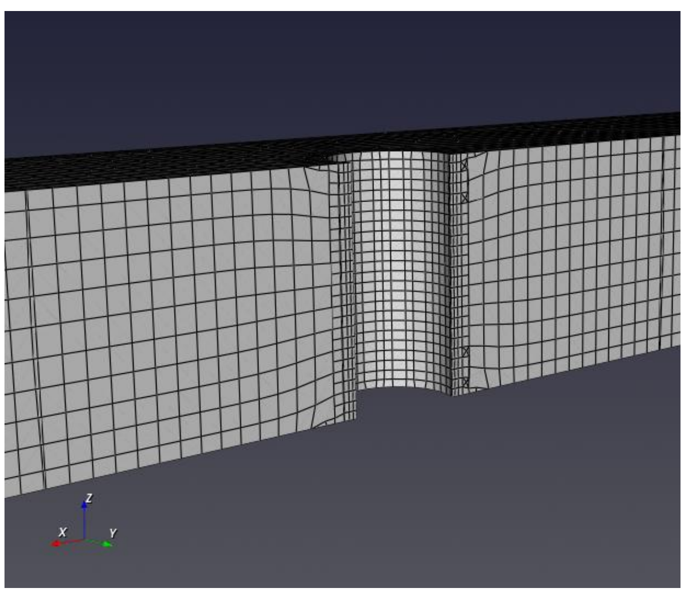

Figure 46. Mesh cut in Y direction.

Click **To Setup**.

Setup

Keep the default settings for **Fluid model** panel. Enable **Heat transfer** in the **Heat model** panel. Set boundary conditions:

- **inlet**: velocity 2 m/s, temperature 288.15 K
- **outlet**: keep default settings
- **wall** (duct walls): set temperature 350 K
- **wall-hot** (rod walls): set temperature 500 K
- **symmetry**: apply symmetry boundary conditions to faces named symmetry.

The model with set boundary conditions shown in Figure 47.

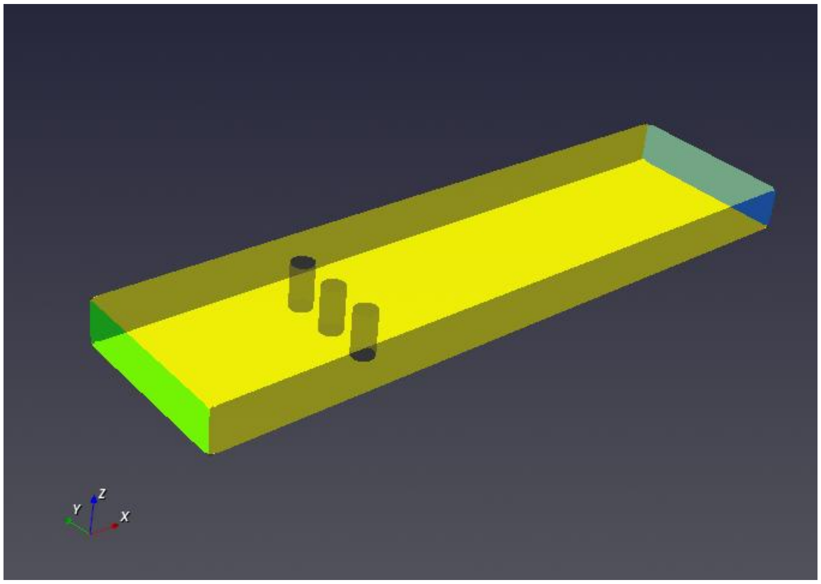

Figure 47. Boundary conditions applied.

Click **To Solution**.

Solution

Create a monitoring point behind the central rod at (0.46, 0.20, 0.05) and monitor temperature **T**. Run the simulation. The converged solution is shown in Figure 48.

Figure 48. Convergence and monitoring graphs.

Click **To Result**.

Result

Create two planes at Y = 0.2 m and Z = 0.05 m and visualize the temperature fields on both planes (Figure 49).

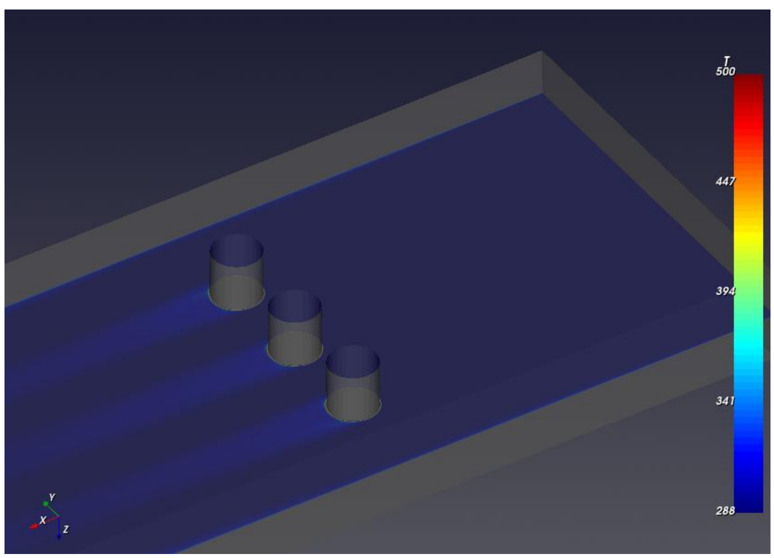

Figure 49. Temperature fields at center horizontal and vertical plane.

On the horizontal plane, visualize **velocity magnitude** and add a **vector plot** to observe separation behind the rods (Figure 50).

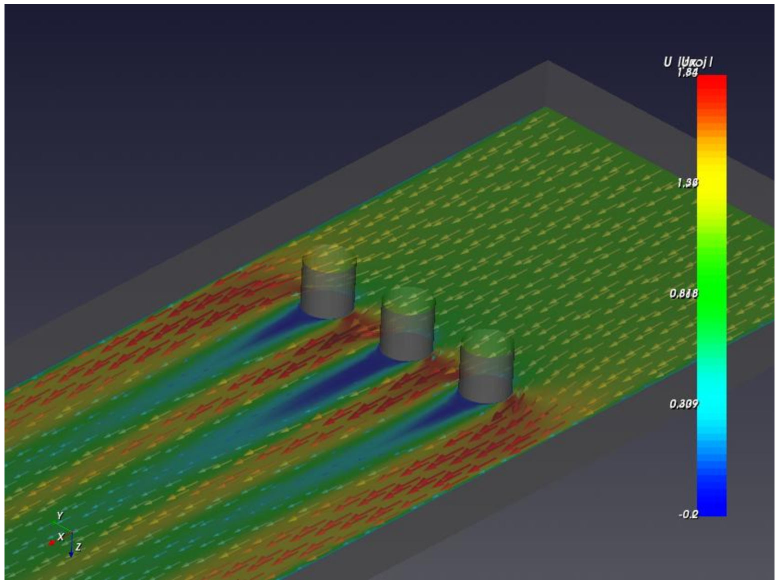

Figure 50. Velocity and vector plot on horizontal plane.

Finally, click **Report** to generate the PDF report.

4. Buoyant flow in a closed box (natural convection)

This case models natural convection in a closed box. The box dimensions are 200 mm (height) × 500 mm (width) × 500 mm (length). A rectangular heat source is located at the center of the box floor. The heat source dimensions are 10 mm (height) × 50 mm (width) × 50 mm (length). Start CFxD from the Windows menu or the desktop icon.

Geometry

Create the points required to build the geometry. The points coordinates are listed in Table 6. Points Pnt 1 and Pnt2 define the box. Points Pnt3 and Pnt4 define the heat source. The units are in mm. In the **Input** panel, use **Unit** button to switch to mm before entering coordinates.

Table 6: Point coordinates (mm).

Point	Coordinates	Comments
Pnt1	(0, 0, 0)	First corner point of the box 1
Pnt2	(200, 500, 500)	Second corner point of the box 1
Pnt3	(0, 200, 200)	First corner point of the box 2
Pnt4	(10, 250, 250)	Second corner point of the box 2

The created points are shown in Figure 51.

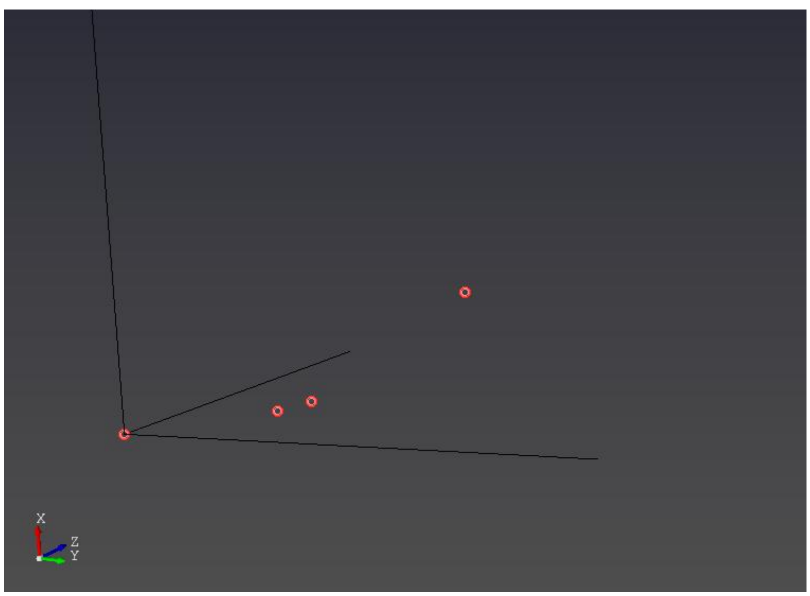

Figure 51. Created points.

Create small and large boxes with Box  tool (Figure 52).

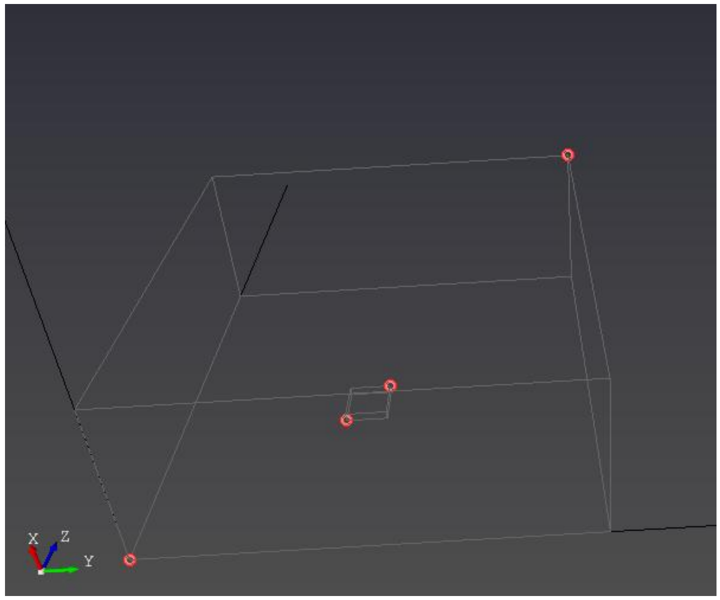

Figure 52. Two boxes.

Extract small box from the large one with Subtract tool. The new Body3 (volume) will be formed (Figure 53).

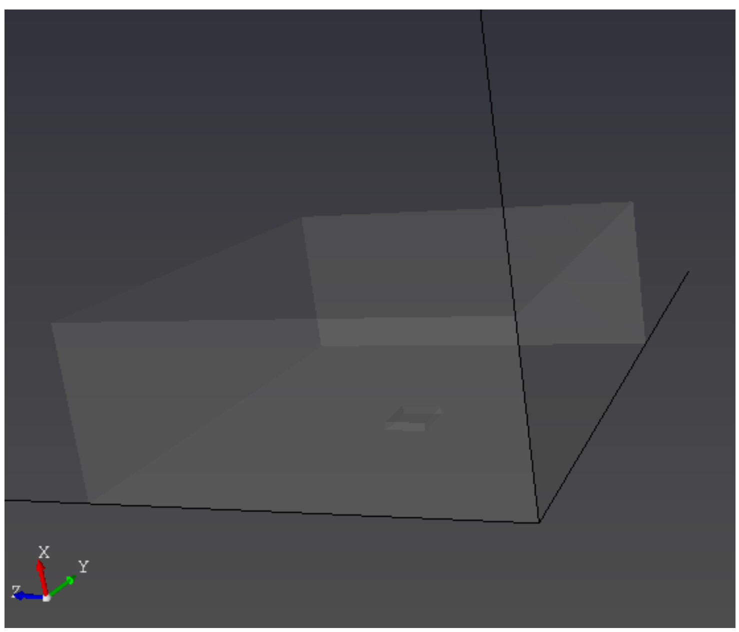

Figure 53. Volume for the simulation.

Add the following names: wall-hot for the heat source and wall-ambient for all other walls. Use Ctrl+RMB for multi-selection. Click **To mesh** step to proceed.

Mesh

Keep the default **Cell size** and set **Refinement** level = 2. Click **Generate**. Inspect the mesh with a cut. Also check the mesh quality plots before proceeding. The generated mesh (cut view) is shown in Figure 54.

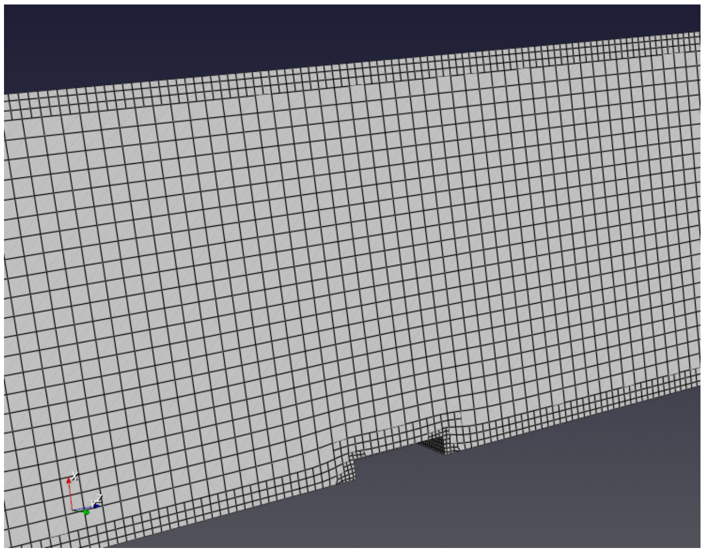

Figure 54. The generated mesh (cut view).

Click **To Setup**.

Setup

Keep the default settings for **Fluid model** panel. Enable **Heat transfer** in the **Heat model** panel. Set temperature 450K for wall-hot and 288.15K for wall-ambient Figure 55.

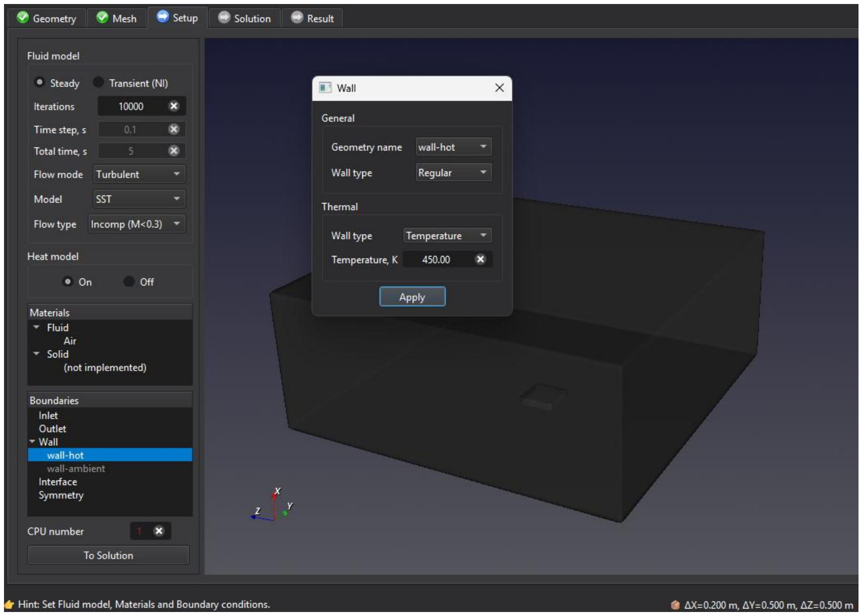

Figure 55. Boundary conditions.

This case does not have inlet and outlet. In the top menu in Settings select the Operating Conditions, enable Buoyancy, keep the default Reference pressure and temperature and set a Gravitational Acceleration in x direction Figure 56.

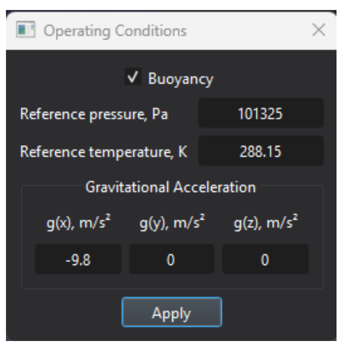

Figure 56. Enable buoyancy.

Click **To Solution**.

Solution

Create a monitoring point above the heat source at (0.200, 0.225, 0.225) and monitor temperature **T**. **Run** the simulation. The converged solution is shown in Figure 57

Figure 57. Converged solution for closed box.

Click **To Result** to move to the next stage.

Result

Create a XZ plane at center Y=0.225m in **Post entities** panel and visualize a vector plot U (Figure 58).

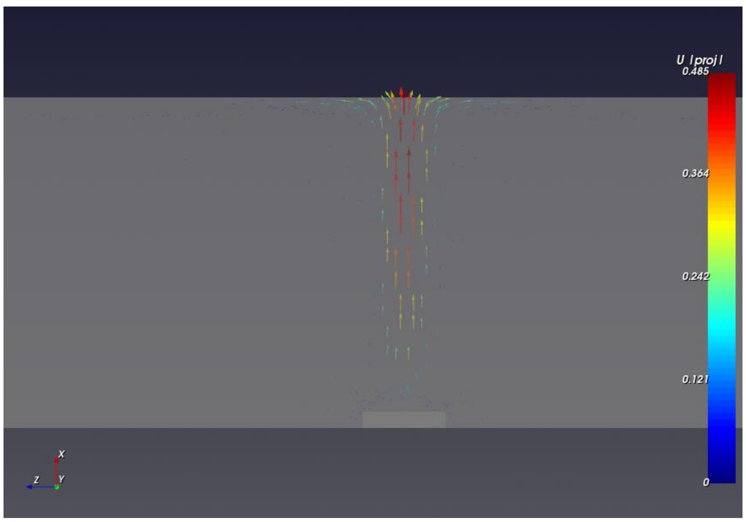

Figure 58. Vector plot on XZ plane at Y=0.225.

Click **Report** to generate the PDF report.

5. Free jet in open air (low Mach compressible flow)

The case represents a free round jet with diameter D = 1.0 m discharging into open air. The jet exit velocity is 300 m/s, while the surrounding (coflow) air velocity is 10 m/s. The jet temperature is 700 K and the surrounding air temperature is 288.15 K (standard ambient). The jet Mach number can be calculated as

$$M = \frac{u}{a} = \frac{u}{\sqrt{kRT}} = \frac{300}{\sqrt{1.4 \cdot 287 \cdot 700}} = 0.57.$$

Since M > 0.3 [1], compressibility effects should be considered.

Geometry

Since the round jet is axisymmetric only a quarter of the configuration will be modelled. The points required to construct the geometry are listed in Table 7 (units: m). Points Pnt1 – Pnt3 define the jet, and points Pnt4 – Pnt6 define the surrounding open-air domain.

Table 7: Point coordinates.

Point	Coordinates	Comments
Pnt1	(0, 0, 0)	Center point of quarter of first circle
Pnt2	(0, 0.5, 0)	Radius point of quarter of first circle
Pnt3	(0, 0, 0.5)	Radius point of quarter of first circle
Pnt4	(5, 0, 0)	Center point of quarter of second circle
Pnt5	(5, 0.5, 0)	Radius point of quarter of second circle
Pnt6	(5, 0, 0.5)	Radius point of quarter of second circle

The created points are shown in Figure 59.

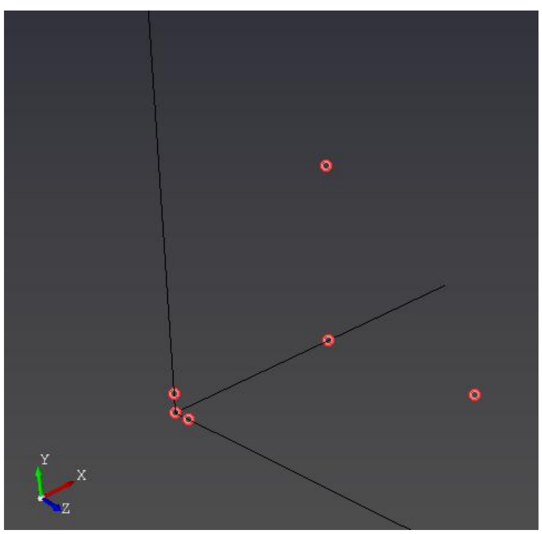

Figure 59. Created points.

Create lines and arc as shown in Figure 60.

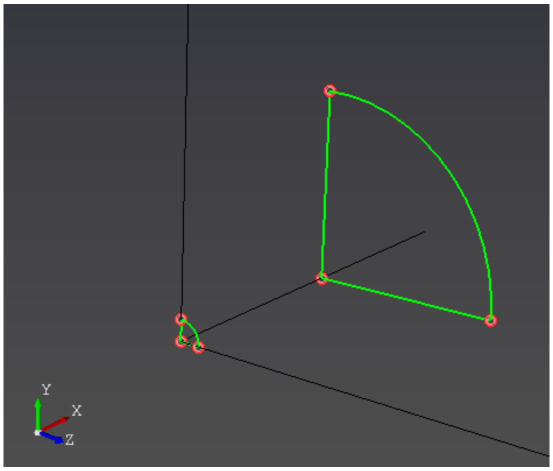

Figure 60. Lines and arcs required for surface creation.

Use **Edge-Surface** creation tool  to generate surfaces by selecting three edges. After the surfaces are created, pull the first (small) surface by 5 m and the second (large) surface by 15 m in the X-direction (Figure 61). During the pull operation, the yellow guide line indicates the extrusion direction. In the example, it points in the -X direction, therefore, to extrude in the +X direction, a negative value must be entered (-15) in the input panel.

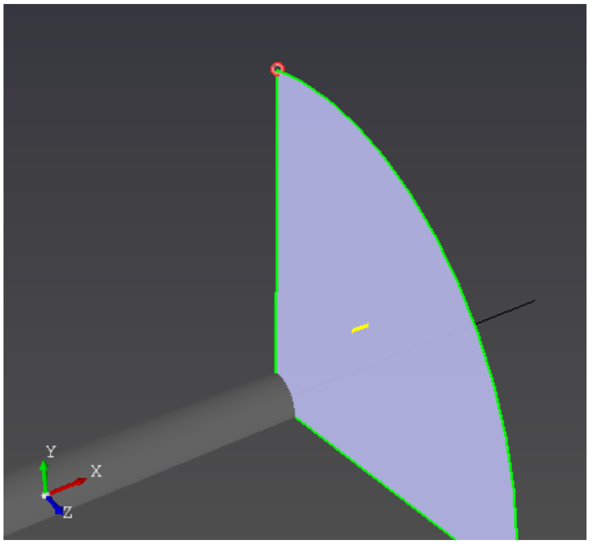

Figure 61. Pull a sector of circle.

Use a Boolean unite operation to combine two bodies. After combining, set the names for faces as shown in Figure 62.

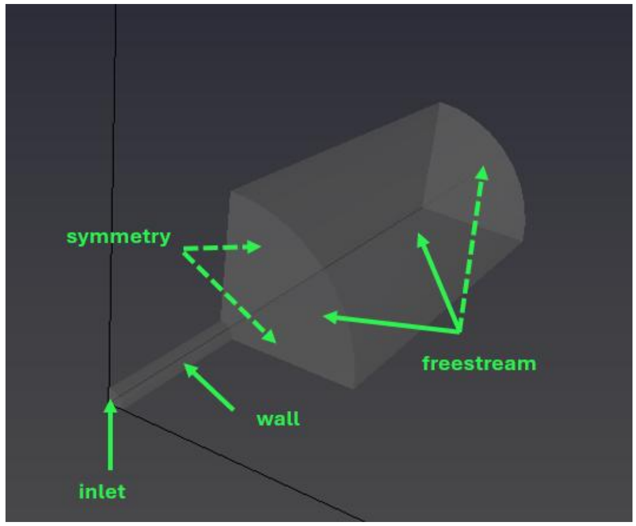

Figure 62. Naming of faces.

Click **To mesh** step to proceed.

Mesh

Keep the default **Cell size** and set **Refinement** level = 2. Click **Generate**. Inspect the mesh with rotation. Also check the mesh quality plots before proceeding. The generated mesh is shown in Figure 63.

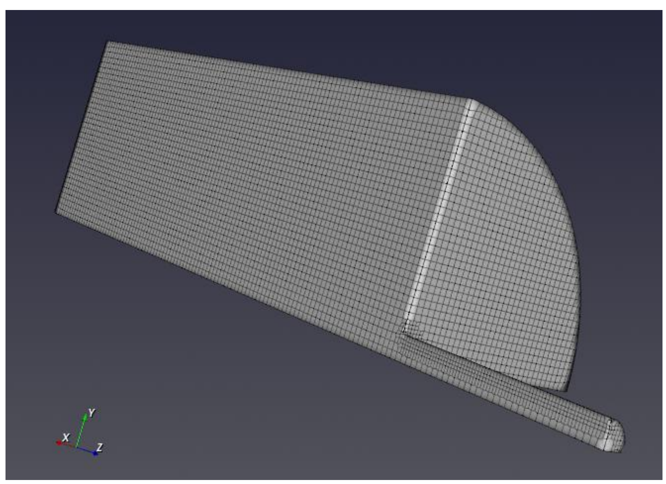

Figure 63. Generated mesh.

Click **To Setup** to move to the next stage.

Setup

In **Fluid model** panel change default SST model to k-eps model. k-eps model is more suitable for free jet. Select **Comp (M<1)** in the **Flow type**. It will automatically activate **Heat transfer** in the **Heat model** panel as the compressible flow modeling requires the energy equation. Set the following boundary conditions:

- **inlet**: velocity 300 m/s, temperature 700 K, turbulent intensity 5%, length scale 0.5 m,
- **freestream**: velocity 10 m/s, temperature 288.15 K, turbulent intensity 1%, length scale 5 m.
- **wall**: set adiabatic
- **symmetry**: apply symmetry boundary conditions to faces named symmetry.

The model with set boundary conditions is shown in Figure 64.

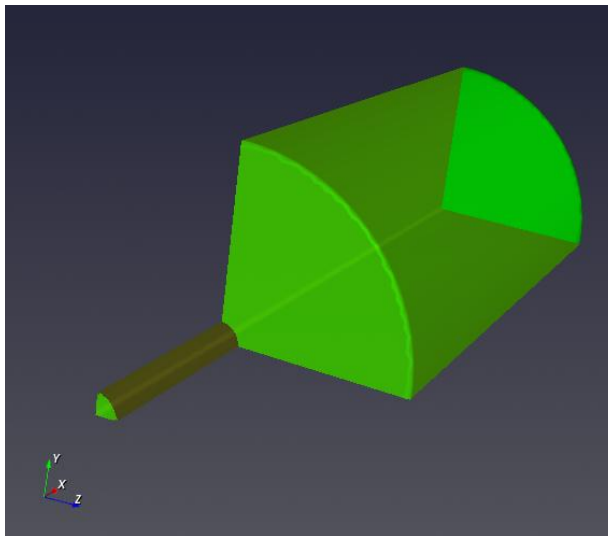

Figure 64. Boundary conditions applied.

Click **To Solution** to move to the next stage.

Solution

Add a monitoring point at 5 m from the jet exit at (10, 0, 0) and select Ux velocity as monitoring variable. Run the simulation. The convergence and monitoring graphs are shown in Figure 65Figure 48.

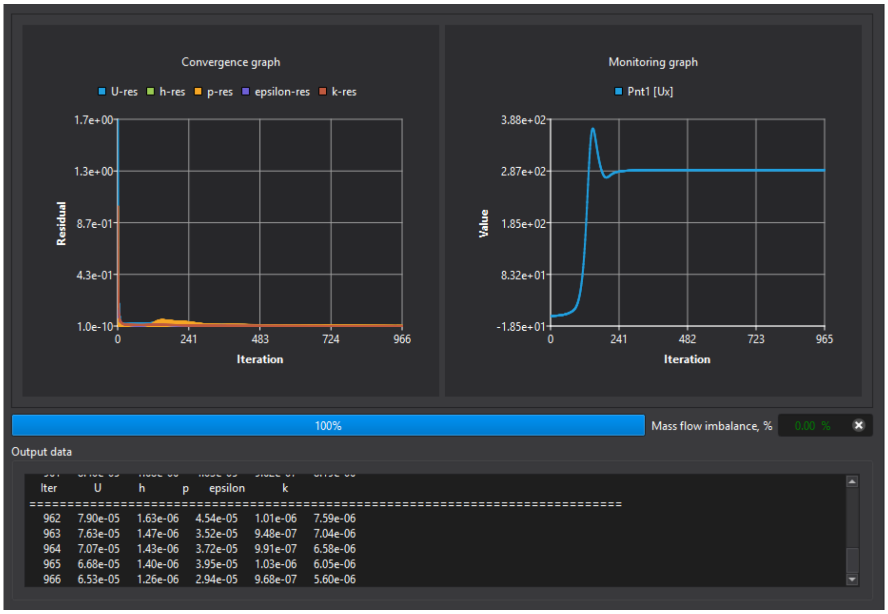

Figure 65. Convergence and monitoring graphs.

Click **To Result** to move to the next stage.

Result

Click on symmetry in **Post entities** panel and visualize Umag (Figure 66).

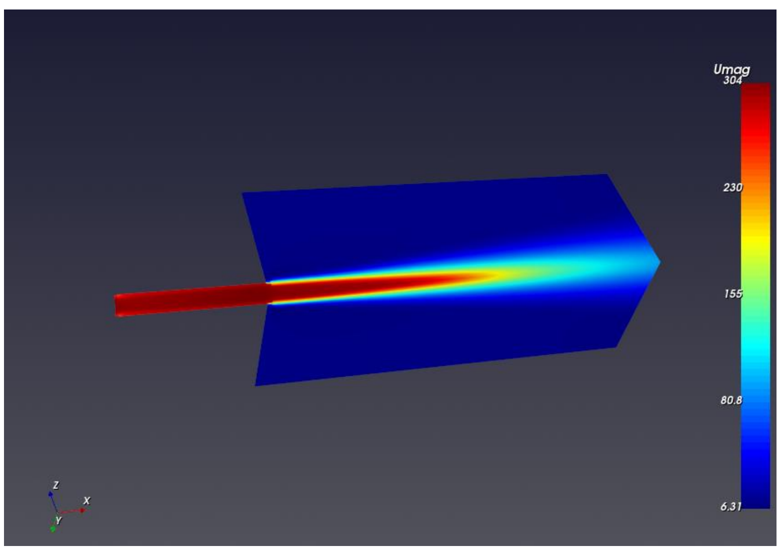

Figure 66. Absolute velocity contour on symmetry planes.

Create a post line with two points (10,0,0) and (10,5,0) and generate a chart for the temperature Figure 67.

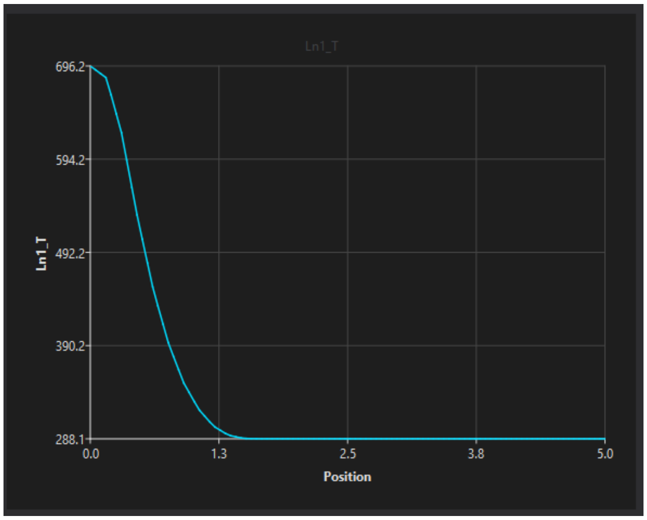

Figure 67. Temperature chart at 5m from jet exit.

Click **Report** to generate the PDF report.

References

1. Software user guide. (Theory and general overview)
2. Yun A. Computational Fluid Dynamics: from zero to guru. Creative Space, 2019.